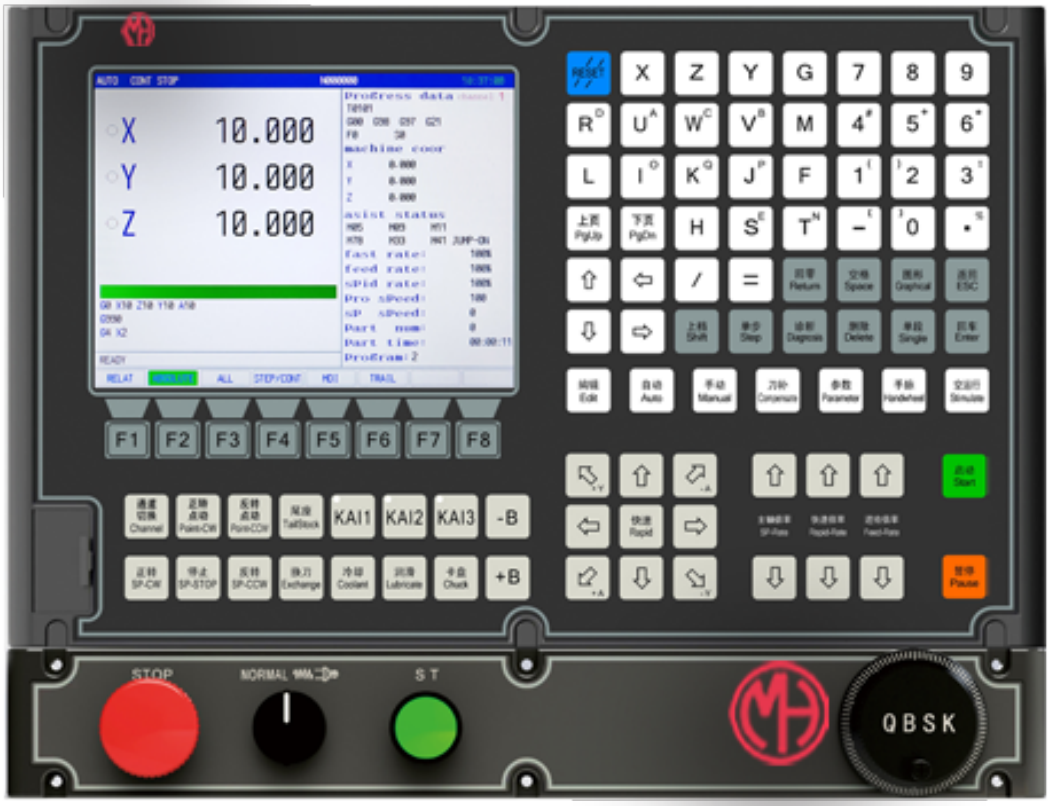
NG98T

catalogue

Introduction to

NG98T,NG98Ta,NG98TT,NG98T-V is the lathe CNC developed and manufactured by our company. This productadopts 8-inch LCD screen real-time monitoring function and provides I/O interface of operation panel. It is the bestchoice for medium and high-grade CNC lathes.



Product technical features:

1. X, Y, Z, A, B, C six-axis control, six-axis linkage, μm-level interpolation accuracy, maximum feed speed 15 m/min, maximum fast speed 30 m/min;
2. Output dual 0V ~ 10V spindle speed signal, which can be adapted to servo spindle or spindle inverter;
3. It has the functions of pitch error compensation, tool tip radius compensation, tool wear compensation and reverse clearance compensation;
4. Support metric/imperial programming, with automatic chamfering and tool life management functions
5, 8 inch 800*600 LCD display, support Chinese display and track graphics display;
6.40/32 point general I/O,
7, 64/64 point operation panel I/O, users can freely configure the operation panel;
8. With real-time clock function and multi-level operation authority management function.
9. The program name and program folder can be named in Chinese or English, or in Chinese and English digital symbols.

Main technical specifications

Movement Control	Control axis: X axis, Z axis, Y axis, A axis, B axis, C axis; Linkage (interpolation) axis
	Axis output signal: pulse + direction and orthogonal pulse, suitable for AC servo or stepper motor drive unit
	Interpolation function: straight line, arc
	Position command range:-9999.999mm ~ 9999.999mm; Minimum command unit: 0.001mm
	Electronic gear: instruction multiplication coefficient is 1~32767 and instruction frequency division coefficient is 1~32767
	Fast moving speed: up to 30000 mm/min Fast magnification: F0, 25%, 50%, 100 four levels of real-time adjustment
	Cutting feed speed: up to 15000 mm/min or 500mm/r (per rev feed) Feed rate: 0~150 16 real-time adjustment
	Manual feed speed: arbitrarily set the F value, or manually enter the 0-9 key to set the F30-F5000
	Hand wheel feed: 0.001mm, 0.01mm, 0.1mm three gears, automatic hand wheel test run program
	Acceleration and deceleration: rapid movement adopts S-type acceleration and deceleration, and cutting feed adopts exponential acceleration and deceleration.
	Automatic chamfering function
G Function	29G codes: G00, G01, G02, G03, G04, G28, G30, G32, G33, G34, G40, G41, G42, G50, G 51, G65, G70, G71, G72, G73, G74, G75, G76, G90, G92, G94, G96, G97, G98, G99, macro instruction G65 can complete 27 arithmetic, logical operations and jumps
Thread Processing	tapping function; single-head/multi-head metric-inch straight thread, taper thread, end thread; variable pitch thread. The length, angle and speed characteristics of thread tail withdrawal can be set, and high-speed tail withdrawal is made. Thread pitch: 0.0001mm ~ 500mm or 0.06 teeth/inch ~ 25400 teeth/inch
	Spindle encoder: the number of encoder lines can be set (100p/r ~ 5000p/r)
	Transmission ratio of encoder and spindle:(1~255):(1~255)
Accuracy Compensation	Backlash compensation: 0mm ~ 2.000mm for each axis
	Pitch error compensation: 512 compensation points, each axis compensation point can be set arbitrarily, each point compensation amount:-0.255~0.255mm
	Tool compensation: X and Z -axis 32 sets of tool length compensation, tool tip radius compensation (compensation method C) tool setting method: fixed-point tool setting and trial cutting tool setting Tool compensation execution method: moving tool or offset coordinate execution tool compensation
M Code	Special M commands (non-redefinable):M02, M30, M20, M97, M98, M99, M, 500,M501,M502,M503,M504,M505,M506,M600,M601,m1800
	Standard configuration M code: M00, M03, M04, M05, M08, M09, M10, M11, M74, M75, M78, M79, M58, M59 M32, M33, M41, M42, M43, M44, M60, M61, M62, M63, M65, M67, M69, M71, M73
T Code	32 cutting positions at most (T01 □□ ~ T32 □□)
spindle speed control	Speed switching value control mode: S□□ command
	Speed analog voltage dual-channel control mode: 0V ~ 10V voltage output, S□□□□ or SS□□□□ given spindle speed (r/min) or cutting linear speed (m/min), can be matched with servo spindle or frequency converter to support 4 gear spindle mechanical gear.
Display Interface	Display: 800 × 600 dot matrix, 8″ liquid crystal display (LCD)
	Display: English, can display the processing trajectory graphics
	Alarm log, real-time clock function
	Softkey operation display
Procedure Edit	Program capacity: 100 M, up to 10000 programs, supports user macro program calls, subprograms quadruple nesting
	Editing method: Program name can be edited with Chinese and English numbers. Full-screen editing, support for relative coordinates, absolute coordinates and mixed coordinates programming

1. Fundamentals of Programming

1.1 machine coordinate system and mechanical zero point

The machine tool coordinate system is the reference coordinate system for CNC to calculate the coordinates. It is the inherent coordinate system of the machine tool. The origin of the machine tool coordinate system is called the mechanical reference point or the mechanical zero point. The mechanical zero point is determined by the zero return switch installed on the machine tool. Normally, the zero return switch is installed at the maximum stroke of the positive direction of the X axis and the Z axis. After the mechanical zeroing operation, the machine coordinate system with the current position as the origin of coordinates is established.

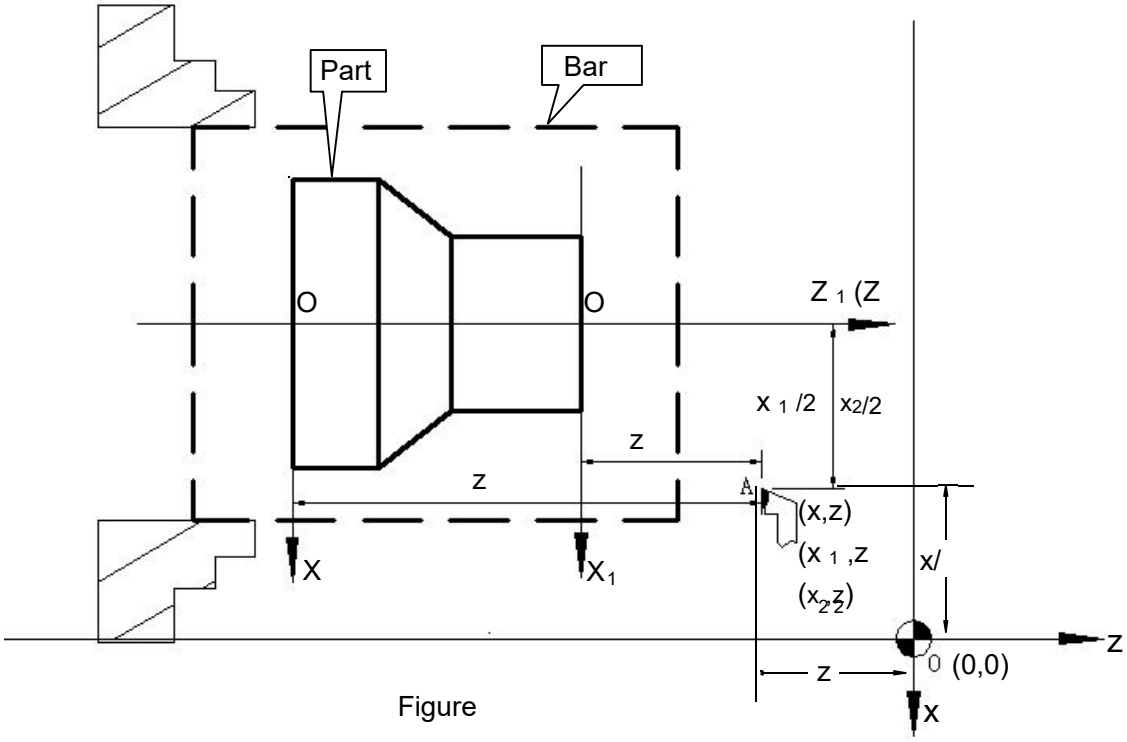
Note: If the zero switch is not installed on the lathe, please do not perform mechanical zero return operation, otherwise it may cause movement beyond the stroke limit and mechanical damage.

1.2 workpiece coordinate system and program zero point

The workpiece coordinate system is a rectangular coordinate system set by the part drawing, also known as a floating coordinate system. When the parts are clamped on the machine tool, the absolute coordinates of the current position of the tool are set with the G50 command according to the size of the workpiece, and the workpiece coordinate system is established in the CNC. Usually the Z axis of the workpiece coordinate system coincides with the spindle axis, and the X axis is located at the head or tail end of the part. Once established, the workpiece coordinate system is valid until it is replaced by a new workpiece coordinate system.

Using G50 to set the current position of the workpiece coordinate system is called the program zero point, and the program returns to this position after the operation of returning to zero.

Note: If the **G50** instruction is not used to set the workpiece coordinate system after power-on**,** please do not perform the operation of returning to program zero, otherwise an alarm will be generated.



Figure

In the figure, X O Z is the machine coordinate system, X_1 O_1 Z_1 the workpiece coordinate system for the X axis at the beginning of the workpiece, X_2 O_2 Z_2 for the X-axis at the end of the workpiece. Piece coordinate system, O is the mechanical zero point, A is the tip, and the coordinates of A in the above three-coordinate system are as follows:

The coordinates of point A in the machine tool coordinate system are (x,z);

The coordinates of point a in the x_1 o_1 z_1 coordinate system are (x_1 ,z_1);

The coordinates of point a in the x_2 o_2 z_2 coordinate system are (x_2 ,z_2);

1.3 interpolation function

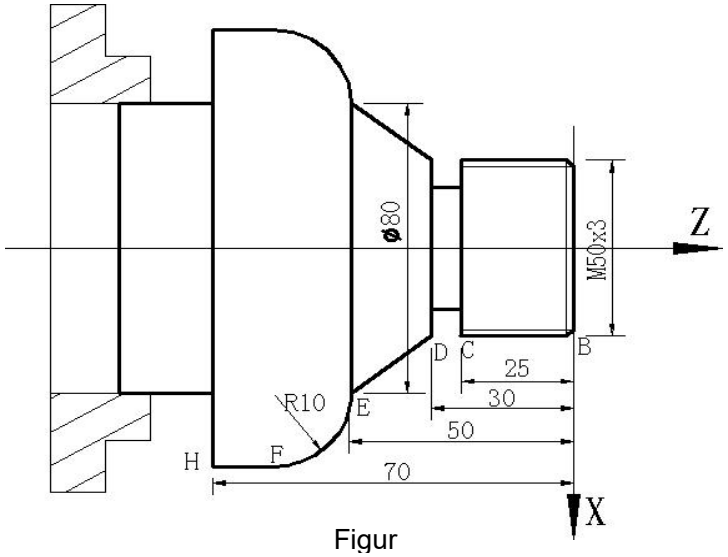
Interpolation refers to the simultaneous movement of 2 or more axes, and the trajectory of the motion synthesis conforms to the determined mathematical relationship to form a two-dimensional (plane) or three-dimensional (space) contour. This motion control method is also called contour control. The motion axis controlled during interpolation is called the linkage axis, and the movement amount, movement direction and movement speed of the linkage axis are controlled at the same time during the whole movement process to form the required synthetic motion trajectory. Only control 1 axis or multi-axis movement end point, do not control the movement process of the movement trajectory, this kind of motion control method is called positioning control.

The X axis and the Z axis are linkage axes, which belong to the linkage of 2 axes, and have straight line, arc and thread interpolation functions.

Linear interpolation: X and Z axis of the synthetic motion trajectory from the start point to the end of a straight line.

Arc interpolation: X and Z axis of the synthetic motion trajectory for the radius specified by R, or by the center of the circle I, K specified from the start point to the end of the arc. Thread interpolation: the angle of rotation of the spindle determines the movement of the X axis or the Z axis or the two axes, so that the tool forms a spiral cutting track on the surface of the rotating body with the spindle to realize thread turning. In the thread interpolation mode, the feed axis follows the rotation of the spindle, the long axis of the spindle rotates one thread cutting moves one pitch, and the short axis and the long axis are interpolated linearly.

Example:



Figur

...

G32 W-27 F3; B→C; thread interpolation

G1 X50 Z-30 F100;

G1 X80 Z-50; D→E; linear interpolation

G3 X100 W-10 R10; E→F; circular interpolation

...

M30;

1.4 absolute coordinate programming and relative coordinate programming

When writing a program, it is necessary to give the coordinate value of the trajectory end point or the target position. According to the type of programmed coordinate value, it can be divided into three programming methods: absolute coordinate programming, relative coordinate programming and mixed coordinate programming.

Programming using the absolute coordinate values of the X and Z axes (expressed in X and Z) is called absolute coordinate programming; programming using the relative displacement of the X and Z axes (expressed in U and W) is called relative coordinate programming;

This system allows the use of absolute programming coordinates and relative displacement programming in the same program segment X and Z axes, respectively, called mixed coordinate programming.

example: A→B linear interpolation.

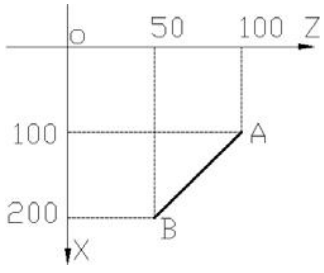


Figure 1-8

Absolute coordinate programming: G01 X200 Z50;
Relative coordinate programming: G01 U100 W-50;
Mixed coordinate programming: G01 X200. W-50; or G01 U100. Z50.;

Note: When there are instruction addresses **X**, **U** or **Z**, **W** in a program segment, **X** and **Z** instruction words are valid.

G50 X10. Z20;
G01 X20. W30. U20. Z30.; [The end coordinates of this program segment are (**X20**, **Z30**)]]

1.5 diameter programming and radius programming

According to the X-axis coordinate value when programming to the diameter value or radius value input can be divided into: diameter programming, radius programming.
Diameter programming: When the comprehensive parameter No. 2 is 1, the command value of the X axis in the program is input according to the diameter value. At this time, the coordinate of the X axis is displayed as the diameter value.
Radius programming: When the comprehensive parameter No. 2 is 0, the command value of the X axis in the program is input according to the radius value. At this time, the coordinates of the X axis are displayed as the radius value.

Table 1-1 Addresses Related to Diameter Programming or Radius Programming Settings

	Address	Description	Diameter Programming	Radius Programming
Addresses related to settings for diameter and radius programming	X	X-axis coordinate	Diameter value representation	Radius value representation
		G50 Set X-axis coordinates		
	U	X-axis movement increment	Diameter value representation	Radius value representation
		X in G71, G72, G73 instructions Shaft finishing allowance	Diameter value representation	Radius value representation
	R	Regression after cutting in G75	Diameter value representation	Diameter value representation
		When cutting to the end point in G74 Regression		

In addition to the addresses listed in Table 1-1, other addresses and data, such as arc radius, G90 taper and other X-axis command values are as follows
The radius value is entered regardless of the setting for diameter programming or radius programming.

Note 1: In the following description of this specification, diameter programming is adopted unless otherwise specified.

1.6 Metric-Imperial Conversion

1. Instruction code

Command format: G20;(inch input)

G21;(mm input)

G21 metric input, the system is in G21 status after power-on;

G20 imperial input, after the system is powered on, it is in G20 status.

	metric input, metric output	Imperial Input, Imperial Output
	One pulse corresponds to 1u	0.0001inch for one pulse
Minimum Input Increment	0.001mm	0.0001inch
Minimum Instruction Increment	0.001mm	0.0001inch
Maximum programming size	± 9999.999mm	± 999.9999inch
Single step/handwheel incremental feed	0.001,0.01,0.1mm	0.0001,0.001,0.01inch
Tool compensation values (X, Z, R)	± 999.999mm	± 99.9999inch
Soft limit of each axis	-99999999um ~ + 99999999um	-999.9999inch ~ +999.9999inch
Minimum value of reverse inter-compensation of each axis	0.001mm	0.0001inch
Minimum value of screw compensation for each axis	0.001mm	0.0001inch
Set value of mechanical zero coordinate of each axis Minimum	0.001mm	0.0001inch
Fixed the amount of retreated, advance of the cycle command. Knife amount, minimum cutting amount minimum	0.001mm	0.0001inch

4. Changes in parameter units

Some values and parameters change after the metric-imperial conversion:

- Value of F-code instruction
- Value of position command
- Knife compensation value
- Handwheel increment, single step increment
- Individual data parameters

All system parameters relate to units of length and speed: the imperial decimal point is shifted left from the metric decimal point by 1. For example, the parameter value of 99999999 118 is 99999.999mm in metric system.

Imperial representation: 9999.9999inch

2 Composition of the procedure

In order to complete the automatic processing of parts, users need to write parts program (hereinafter referred to as program) according to the instruction format of CNC. CNC executive program to complete the machine tool feed movement, spindle start and stop, tool selection, cooling, lubrication and other control, so as to achieve the processing of parts.

Program example:

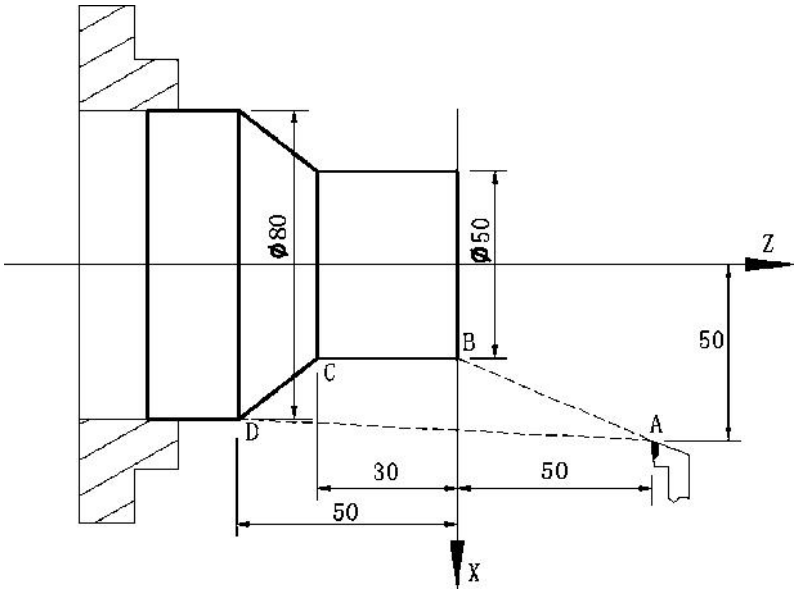


Figure 1-9

NEW9	;	(Program Name)
N0005	G0 X100 Z50;	(Quickly locate to point A)
N0010	M12;	(Clamping the workpiece)
N0015	T0101;	(Change the No.1 knife and execute the No.1 knife deviation)
N0020	M3 S600;	(Start the spindle and set the spindle speed to 600 r/min minutes)
N0025	M8	(Coolant on)
N0030	G1 X50 Z0 F600;	(at 600 mm/min speed close to B point)
N0040	W-30 F200;	(Cut from point B to point C)
N0050	X80 W-20 F150;	(Cut from point C to point D)
N0060	G0 X100 Z50;	(Fast back to point A)
N0070	T0100;	(Cancel knife deviation)
N0080	M5 S0;	(Stop spindle)
N0090	M9;	(Turn off coolant)
N0100	M13;	(Loosen the workpiece)
N0110	M30;	(End of the program, turn off the spindle and coolant)
N0120	%	

After performing the above procedure, the tool will go out of the A→B→C→D→A trajectory.

Up to 1000 programs can be stored, in order to identify and distinguish each program, each program has a unique program name (program name is not allowed to repeat), the program name is located at the beginning of the program Chinese, numbers, letters and the first three mixed.

Instruction Word

The instruction word is the basic instruction unit used to command the CNC to complete the control function. The instruction word consists of an English letter (called the instruction address) and a value after it (called the instruction value, which is a signed number or an unsigned number). The instruction address specifies the meaning of the subsequent instruction value, and the same instruction address may have different meanings in the case of different combinations of instruction words. Table 1-2 is a list of all instruction words.

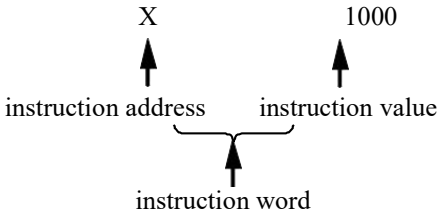


Table 1-2 List of Instruction Words

Instruction Address	Command value range	functional significance
N	0 to 9999	Program Section Number
G	00 to 99	Preparation function
X	-9999.999~9999.999(mm)	X-axis coordinate
	0~9999.999(s)	Pause time
Z	-9999.999~9999.999(mm)	Z-axis coordinate
U	-9999.999~9999.999(mm)	X-axis increment
	0~9999.999(s)	Pause time
	-99.999~99.999(mm)	X-axis finishing allowance in G71, G72, G73 commands
	0.001~99.999(mm)	G71 middle cutting depth
	-9999.999~9999.999(mm)	X-axis retreating distance in G73
W	-9999.999~9999.999(mm)	Z-axis increment
Instruction Address	Command value range	functional significance
	0.001~99.999(mm)	G72 middle cutting depth
	-99.999~99.999(mm)	Z-axis finishing allowance in G71, G72, G73 instructions
	-9999.999~9999.999(mm)	Z-axis retreating distance in G73
R	-9999.999~9999.999(mm)	arc radius
	0.001~99.999(mm)	G71, G72 cycle withdrawal
	1~9999 (times)	Number of Rough Cycle in G73
	0.001~99.999(mm)	Regression after cutting in G74, G75
	0.001~99.999(mm)	Regression when cutting to the end point in G74 and G75
	0.001~9999.999(mm)	G76 Medium Finishing Allowance
	-9999.999~9999.999(mm)	G90, G92, G94, G96 middle taper
I	-9999.999~9999.999(mm)	The relative starting point of the arc center is in the X-axis vector
	0.06~25400 (teeth/inch)	English thread number
K	-9999.999~9999.999(mm)	The relative starting point of the arc center is in the Z-axis vector
F	0~8000(mm/min)	sub-feed speed
	0.0001~500(mm/r)	Transfer feed speed
	0.001~500(mm)	Metric thread lead

S	0~9999(r/min)	Spindle speed designation
	00-04	Multi-gear spindle output
T	01 to 32	Tool function
M	00 to 99	Auxiliary function output, program execution process
	9000~9999	subroutine call
P	0 to 9999999(0.001 seconds)	Pause time
	0 to 9999	Number of called subroutine
	0 to 999	Number of subroutine calls
	0~9999999(0.001mm)	X-axis cyclic movement in G74, G75
		Thread cutting parameters in G76
	0 to 9999	Start program segment number in compound cycle instruction finishing program segment
Q	0 to 9999	End program segment number in compound cycle instruction finishing program segment
	0~9999999(0.001mm)	Z-axis cyclic movement in G74 and G75
	1~9999999(0.001mm)	First cut-in in G76
	1~9999999(0.001mm)	Minimum cut-in in G76
	0 to 360000	G32 Starting angle, Z signal and thread cutting starting point of spindle Offset angle.
H	01 to 99	Operators in G65

nt, except for addresses such as n, g, s, t, h, l, etc., other addresses can only appear once, otherwise an alarm will be generated (instruction words are repeated in the same program segment). When N, S, T, H, L instruction words are repeatedly input in the same program segment, the last instruction word of the same address is valid. When G instructions of the same group are repeatedly entered in the same program segment, the last G instruction is valid.

Program Section Number

The program segment number consists of the address n and the following 4 digits: N0000 ~ N9999, and the leading zero can be omitted. The program segment number should be at the beginning of the program segment, otherwise it is invalid.

The program segment number may not be entered, but the target program segment called or jumped by the program must have a program segment number. The sequence of program segment numbers can be arbitrary, and their intervals can also be unequal. In order to facilitate the search and analysis of programs, it is recommended that the program segment numbers increase or decrease in the programmed order.

If the 115 number is set to "automatic sequence number" and "insert" in the comprehensive parameter, an incremented program segment number will be automatically generated when the program segment is inserted.

block selector

If you want to not execute a 1 program segment during program execution (but do not want to delete the program segment), insert ";" This is when the program is executed.

The program segment will be skipped and not executed.

Procedure Notes

In order to facilitate users to find programs, each program can edit program comments with no more than 20 characters (10 Chinese characters). The program comments are located in the program name, and only English letters and numbers can be used to edit the program name on CNC. The program name can be edited in Chinese on the computer. After the program is downloaded to CNC, CNC can display the Chinese program name.

2.1 main program and subprogram

In order to simplify programming, when the same or similar machining trajectory and control process need to be used multiple times, the program instructions of this part can be edited as an independent program to call. The program that calls the program is called the main program, and the called program (**the subroutine name must start with "o" and be followed by 4 digits, for**

example: O0008 (programs are not allowed to be written as O8) end of M99) is called a subroutine. The subprogram takes up the program capacity and storage space of the system as the main program. The subprogram must have its own independent program name. The subprogram can be called by any other main program or run independently. After the subroutine ends, it returns to the main program to continue execution, as shown in the figure below.

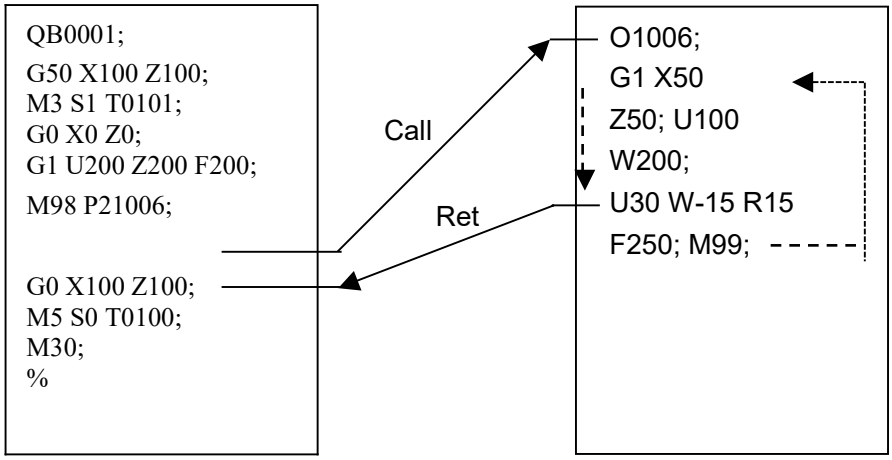


Figure 1-3

3 The operation of the program

3.1 the order in which the program runs

The currently open program must be run in automatic mode, and two or more programs cannot be opened at the same time. Therefore,

Only one program can run at any one time. When opening a program, the cursor is located at the beginning of the first program segment, and the cursor can be moved in the edit operation mode. In the operation stop state of the automatic operation mode, the operation of the program is started from the program segment where the current cursor is located by the cycle start signal, usually one by one according to the sequence of the program segment, until the M02 or M30 instruction is executed, and the program operation stops. The cursor moves as the program runs and is always at the beginning of the current program segment. The order or state in which a program runs changes when:

- When the program is running, press the 

reset key or the emergency stop button to terminate the program;
- CNC alarm is generated when the program is running, and the program is terminated;
- When the program is running, the operation mode is switched to the editing operation mode. You can view the program content, but you cannot modify the program.
- When the program is running, the operation mode is switched to the knife compensation operation mode. You can view and modify the knife compensation content, but the next time you run the program, it will take effect.
- When the program is running, the operation mode cannot be switched to manual, parameter, handwheel, single step and program return operation mode.
- When the program is running, press the pause key, the program is suspended, and when the start key is pressed again, the program continues to run from the stopped position;
- When the single-segment switch is turned on, the program operation will be suspended after the end of each program segment. You need to press the start key again to continue running from the next 1 program segment;
- The circuit breaker is open, and the program segment with ";" in front of the program segment is skipped and not executed;
- When executing the G65 jump instruction, turn to the jump target program segment to run;
- The program execution sequence of G70 ~ 73 compound loop instruction is quite special, see the "G instruction" in this article;
- When executing M98 or M9000 ~ M9999 instructions, call the corresponding subroutine or macro program to run; when the subroutine or macro program is finished, when executing M99 instruction, return to the next 1 of the calling program segment in the main program to run (if M99 instruction specifies the target program segment number to be returned, turn to the target program segment to run);
- When the M99 instruction is executed in the main program (the operation of the program is not started by the call of other programs), the first segment of the program is returned to continue to run, and the current program will run repeatedly.

3.2 the order of execution of instruction words within a program segment

A program segment can have multiple instruction words such as G, X, Z, F, R, M, S, T, etc., and other instruction words are directly processed by CNC. M98,

M99, M9000 ~ M9999, and the S command word for giving the spindle speed in rev/min and m/min are also directly processed by NC.

When the G instruction and M00, M01, M02, M30 are in the same program segment, NC executes the G instruction before executing the M instruction.

When the g instruction word and M98, M99, M9000 ~ M9999 instruction words are in the same program segment, NC executes the g instruction before executing it

These M instruction words.

4 Keyboard Description

4.1Increase or decrease of magnification

(1) Fast magnification

The fast magnification has a total of 4 gears from 0% to 100. Adjust the manual fast feed speed through the fast magnification key. The speed after the magnification is adjusted to the lowest gear of 0% is set as the transmission parameter No. 98.

(2) Feed rate

The feed ratio is 0% ~ 150 in a total of 16 gears. Adjusted by the feed ratio key, the feed ratio is valid for the following instructions: G01, G02, G03, and fixed cycle feed F are valid; It is also effective for manual operation.

(3) Spindle magnification

The spindle magnification has eight gears of 50% ~ 150. It is adjusted by the spindle speed magnification key, and the spindle magnification is effective for the first spindle speed S. The lowest level is 50%.

4.2 the use of three-stage pause switch

(1) left: normal processing.

(2) medium: manual feed stop, automatic feed pause. Return to normal when left.

(3) Right: automatic feed pause, spindle M03 pause, when in non-axis operation spindle is not controlled.

4.3See the table below for other keys

letter key	ABCDEFGHIJKLMNOPQRSTUVWXYZ for program commands and parameter settings;
Number Keys	0123456789.-for data input and parameter setting;
Symbol Key	# *[]()/=% for macro program editing;
arrow keys	↑↓ → ← in the programming area is used to move the cursor; In the manual control area, the ↑↓→← A- A + B- B + C- C is used for manual feeding;
Edit Key	"Delete, previous page, next page" is used for program editing;

Functio n keys	"Reset" autorun program Reset End program run and return to program The switch value of the head and spindle is closed, and the operation of the feed axis is reset and stopped; "Return" is used for operation cancellation and function return; "Confirm" is used for function selection and input confirmation; "Parameter" enters the parameter setting menu; "Edit" enters the program editing menu; "Tool Compensation" corrects the tool change error; "Manual" manual function selection; "Automatic" automatic function selection; "Handwheel" starts the handwheel function; "Diagnosis" inputs and outputs diagnosis; "Single segment" selects automatic execution mode, which can be stopped during processing; "Single step" is used for manual single step (I. e. increment) selection; "Graphics" is used for automatic coordinate or graphics processing mode selection; "dry run" is used for automatic processing or automatic dry run mode selection;
Control key	"forward rotation, reverse rotation, stop" is used for the forward rotation, reverse rotation, stop of the spindle; "inching" spindle forward and reverse inching;
	"Cooling" is used to start and stop the coolant; "Return to zero" is used to return X and Z to the zero point of the "Tool change" is used for the transposition of electric knife holder or "Chuck" is used to clamp and loosen the spindle chuck; "thimble" is used to tighten and loosen the spindle thimble; "Lubricate" switch for lubrication; "Fast" is used for manual continuous high and low speed switching; QUICK ↑, QUICK ↓ is used for G00 or manual quick trim; "feed ↑, feed ↓" is used to adjust the feed speed; "Spindle ↑, spindle ↓" is used to adjust the spindle speed; "Start" to execute automatic machining program; "Pause" pauses during automatic machining;
Control	Emergency stop: stop all actions of the machine tool and wait for the Three-stage switch: suspend the lead screw and spindle during Program operation: the implementation of automatic processing

4.4 manual operation

4.4.1 Manual key operation

(1) "F" key: set the manual feed speed, in mm/min, and its input range is: 1-30000 mm/min. For its input method, see Data Input Method in Parameters.

(2) "single step" key: "manual continuous", "manual single step" cycle switch.

(3) "S" key: set the first spindle speed. The spindle speed range is 0-30000, and its maximum value depends on the value of parameter No. 107-110 in the transmission parameter state.

"SS" key: set the second spindle speed. The spindle speed range is 0-30000, and its maximum value depends on the value of parameter No. 111 in the transmission parameter state.

(4) "I" key: modify the single step value in the manual single step mode.

(5) "feed ↑" key: when the comprehensive parameter 190 parameter is set to 0, the feed speed magnification will be adjusted and increased by 10% every time the "feed ↑" key is pressed once. The range is 0-150%, a total of 16 gears.

(6) "Feed ↓" key: when the axis parameter No. 190 parameter is set to 0, the feed speed magnification is reduced by 10% every time the "Feed ↓" key is pressed. The range is 0-150%, a total of 16 gears.

(7) "Quick ↑" key: Each time the "Quick ↑" key is pressed, G00 or manual quick magnification trimming will be increased by 25%. Range 0-100% total 6 4 gears

(8) "Fast ↓" key: Each time the "Fast↑" key is pressed, G00 or manual fast magnification adjustment will be reduced by 25%. The range is 0-100% total 5.

(9) "Spindle ↑" key: when the comprehensive parameter No. 189 is set to 0, the speed magnification of the first spindle will be increased by 10% each time the "Spindle ↑" key is pressed. The range is 50-120% for a total of eight gears.

(10) "spindle ↓;" key: when the comprehensive parameter No. 189 parameter is set to 0, the first spindle speed magnification is reduced by 10% every time the "spindle ↓" key is pressed. The range is 50-120% for a total of eight gears.

(11) "handwheel" key: select "hand pulse", press the feed rate key can be "0.001", "0.01", "0.1" gear cycle switch.

(12) "Lubrication": The "Lubrication" key is the lubrication on and off repeated key, corresponding to commands M32 and M33.

(13) "automatic" key: select "automatic mode".

(14) "Manual" button: Select "Manual Mode".

(15) Spindle control: "forward rotation, reverse rotation and stop" respectively control the forward rotation, reverse rotation and stop of the spindle, corresponding to instructions M03, M04 and M05.

Press the "jog" key to output a jog signal, that is, when pressed, the spindle rotates, and when lifted, the spindle stops.

(16)"Cooling": The "Cooling" key is the coolant on and off key repeatedly, corresponding to commands M08 and M09.

(17) "Chuck": The "Chuck" key is the key for opening and closing the chuck repeatedly, corresponding to instructions M10 and M11.

(18) Manual speed: press "0", "1", "2", "3", "4", "5", "6", "7", "8", "9" to do "F30", "F60", "F120", "F250", "F500", "F1000", "F1500", "F2000", "F2500", "F3000" manual speed setting.

(19 "tailstock": the "tailstock" key is the tailstock on and off key repeatedly, corresponding to instructions M79 and M78.

(20) Switching between continuous and single step: When the "single step" key is pressed, the manual continuous mode is switched to the single step mode. If it is a single step, the single step value I = XXXX.XXX is displayed.

(21) Return to the mechanical zero point of the machine tool: after pressing the "Return to Zero" key, press the manual direction key of the corresponding axis to automatically return to the zero point of the machine tool of this axis; If you press the "Reset" key, it will stop returning to zero. .

(22) Tool change: press the "tool change" key, and if the tool holder is arranged, it will automatically switch to the next tool; If it is an electric tool holder, turn to the next tool position and stop. Press the "knife change" key to compensate for the knife number of that knife.

If you directly press the "T" key and directly input the tool number and tool compensation number to be changed, the tool holder will be changed to the input tool position number and tool compensation will be performed. See the programming T function for the method.

(23) Coordinate feed: directly press the "↓↑ →←" key to feed in the positive and negative directions of the X-axis and Z-axis respectively.

(24) Speed switching: during continuous feeding, press the "fast" key again, and the flashing display of "fast" on the top of the display screen will immediately convert it into the execution speed corresponding to the 117 number parameter of the system transmission parameter, and release the "fast" key to continue feeding at the original speed. If the set speed is higher than the speed in the parameter, feed at the set speed.

(25) Multi-coordinate display switch, press the "relative coordinates", "absolute coordinates", "integrated coordinates" key can be in the "absolute coordinates", "relative coordinates" "integrated coordinates" 3 interface switch

(26) The number of processed parts is cleared: press the "Delete" key under the automatic interface, and then press the "Confirm" key to clear the number of processed parts.

(27) zero processing time: automatic interface press "T", key, and then press the "confirm" key, processing time reset

(28) Incremental coordinates: select the "relative coordinates" state and press the "U,V,W,A,B .C" keys to set the relative coordinates to 0 respectively;

(29) Machine tool coordinate clearing method: 1. Equipped with incremental system: Press "Delete" in the zero return state, then press "Corresponding Axis" key, and then press Enter key. 2. Absolute value matching system: multiply the positive and negative opposite values of the machine tool coordinate value input by 1000, and then write the corresponding comprehensive parameters 205 number (X axis),206 number (Y axis),207 number (Z axis),208 number (A axis),209 number (B axis) and 210 number (C axis)

Note: The machine coordinates cannot be deleted in the zero-back state when the absolute value function is turned on.

4.4.2 Manual continuous

Continuous operation is based on the press time of the key, press it to feed, lift it to stop feeding. Use the "↓→ ←" key on the panel to feed in the specified axis and direction, and the feed speed is the displayed speed (F data) multiplied by

the speed trim factor value.

During continuous execution, if the "fast" key is pressed, the speed will be switched to the execution speed corresponding to the 117 parameter of the transmission parameter. The actual feed speed is the speed of the 117 parameter value in the transmission parameters multiplied by the fast or feed trim magnification value.

In order to facilitate the user to perform single-axis cutting in the manual function, the manual speed can be set in the manual. When the "F" key prompts F, type the speed value.

When the feed exceeds the hard limit points in the positive and negative directions of the running axis, the feed will be stopped, and it will be prompted that no further feed can be carried out in the direction of the limit axis. At this time, press the reset key and then feed in the opposite direction.

The manual maximum speed is set by the 117 parameter of the transmission parameter. When the set speed is higher than this value, the displayed value is out of range.

4.4.3 Manual Step

This operation is based on the set single step value, each time you press the feed key coordinates to run a single step value. After entering the single-step value operation, the prompt "I = 0.001" indicates that the single-step value is 0.001mm. You can press the "I" key to modify the single-step value and press the "Confirm" key to confirm. Its running speed is the display speed (F data) multiplied by the speed trimming magnification value.

4.4.4 Return to the zero point of the machine tool

Specific methods are as follows:

There is a back-to-zero switch: the back-to-reference point operation is to make each coordinate axis sequentially feed to the base point switch position on the machine tool. For example, when the mechanical zero mode in comprehensive parameter No. 52 is set to 1, and each axis coordinate axis detects zero as a signal and then returns to detect Z pulse signal (whether to check Z pulse is determined by comprehensive parameter No. 23), the machine coordinate data is automatically set to 0.

After the power of the numerical control machine tool is turned on again after the power supply of the numerical control system, the over-travel alarm is released and the emergency stop button is released, it is necessary to return to the reference point again to establish the correct machine coordinate system.

1. The type and mode of the detection signal for returning to zero can be set through the transmission parameter P27, so that only the reference point switch can be detected, and the motor Z pulse signal can be detected after the reference point switch is detected (with higher precision). The motor Z pulse can be detected in the forward direction or in the reverse direction.
2. The direction of returning to zero can be set by comprehensive parameter No. 20 (bit parameter).
3. The level type of the zero switch can be set by the comprehensive parameter No. 7 (bit parameter).
4. The coordinate setting after returning to zero can be set through the transmission parameter 130-35, that is, the coordinate can be quickly moved to the parameter value after returning to zero.

Special attention: Each time the system is started, a reference point return operation must be performed to ensure the accuracy of the machine tool. Due to accidental power failure or abnormal shutdown after starting the system must also be back to zero operation, otherwise it may cause failure.

4.4.5 Hand pulse generator (handwheel)

The handwheel in this system can be configured with panel-type and hand-held handpulse, which is set by comprehensive parameter No. 1.

Panel Handwheel Operation:

Press the "Hand Pulse" key to select "Handwheel Mode". Then press the manual direction key of each axis to select the corresponding axis, and then press "feed rate" to select pulse rate * 1,* 10 and * 100.

Handheld hand pulse operation: select "hand pulse mode" by pressing "hand pulse" key ". Then operate the axis selection switch of hand-held hand-pulse to select the corresponding axis, and operate the axis and magnification switch of hand-held hand-pulse to select the pulse magnification.

Automatic handwheel operation: under the automatic interface of the system, press the hand pulse key, and the system prompt: enter the automatic handwheel mode? Press the enter key to enter the automatic handwheel, press the return key to cancel, after the program presses the start key, the handwheel shakes, the program runs, the handwheel does not shake, the program does not move, the MST command is not controlled by the handwheel

Note: The hand wheel is mainly used for knife, and its running speed is related to the feed rate of each frame of the hand wheel and the speed of the rotating hand wheel.

If the system is equipped with a stepper motor, the feed speed should not be too fast. The rotation speed of the hand-cranked pulse generator should be lower than 200r/min (the hand wheel with 100 pulses per revolution). The

hand wheel mode is invalid in the automatic graphics mode and can only be effective in the coordinate operation mode.

4.4.6 Knife set

When using multiple tools for processing, because the length of each tool is different, it is necessary to determine the tool compensation value between them, that is, to work on the tool. The tool is actually moving the tool to a point on the surface of the workpiece, measuring the actual value of this point directly into the system, the system automatically calculates its deviation, and automatically saves to the corresponding tool offset register.

Press the "Knife" key, then press "X" or "Z", and then press the "Confirm" key to exit after the current axis is right. There are two ways to knife:

Method A (recommended):

(1) Clamp the workpiece, select the appropriate spindle speed and feed speed, and start the spindle.

(2) Select the right tool, such as T0101.

(3) the use of manual continuous way on the workpiece on the car to 1 a small section of the outer circle or inner hole.

(4)Z axis exit (X axis can not move), close the spindle.

(5) Measure the diameter of the workpiece (outer circle or inner hole).

(6) Press the "Try to cut and set the knife" key, press the "X" key to confirm, then enter the above measured value into the dialog box, and press the "Confirm" key to confirm

(7) In the same way, the end face of the workpiece is turned with a tool.

(8) Measure the distance between the end face of the workpiece and the chuck of the spindle (Z = 0).

(9) Press the "Try to cut and set the knife" key, press the "Z" key, then enter the upper measured value into the dialog box, and press the "Confirm" key to confirm.

At this time, T02 second tool has been good. Repeat 1)-9) Other tools can be properly set.

4.5 automatic operation

Automatic refers to the currently edited workpiece program for processing. The system can be started at any point, can also be any line start or any tool start. Any line or any point start must be programmed with absolute coordinates in the program. The automatic mode cannot move the coordinates manually.

Run program selection: under the program interface, press "↑" "↓" to move the cursor to the name of the program to be executed, and press the "Run Program" key to select the program to be run automatically.

Press the "Auto" key in the manual mode to enter the automatic mode.

Multi-coordinate display switch. Press the function key of "Absolute Coordinates", "Relative Coordinates" and "Comprehensive Coordinates" to switch 3 interfaces of "Absolute Coordinates", "Relative Coordinates" and "Comprehensive Coordinates.

4.5.1 Automatic processing method

Selection of "single segment/continuous": press the "single segment" key to switch the single segment/continuous processing mode repeatedly. "Continuous operation" means that after pressing the "run" button, the program executes each program segment in turn until the program ends or a stop instruction appears. "Single-segment operation" is to stop after running only the current line of the program while running the program, waiting for the key operation or press the Run button again. That is, press the "run" button once to run a program. Selection of "Coordinate/Graph": Press the "Graph" key to switch the coordinate/graph execution mode repeatedly. "Automatic coordinates" means that the coordinates of the axes of the program are displayed numerically as coordinates. "Automatic dry run" means that the program runs in a dry run mode and the axes do not move. "Automatic graphics" means that the axes of the program run are displayed in the form of a machining trajectory.

4.5.2 Program start

Start the program must press the "automatic" key to switch to automatic mode, there are the following 2 ways, the program automatically start execution.

(1) Panel press the "start" key.

(2) Connect the RUN signal for external operation.

4.5.4 Program Stopping

There are five ways to stop a program:

(1) Program instructions M00, M02, M30.

(2) Press the "Single Section" button to stop after the current program section runs.

(3) The three-stage switch is placed in the middle or right.

(4) Turn on the external stop HALT signal or press the "pause" key to pause the program.

(5) Press the "RESET" key to stop all actions of the program (such as workbench, spindle, tool holder, etc.)

4.5.5 Real-time control in automatic machining

(1) Three-stage pause switch: used to pause the coordinate feed and stop the spindle. Divided into left, middle and right three gears. When the switch points to the left: coordinate feed and spindle start are not limited. When the switch points to the middle: the axis stops feeding and the spindle stops. When the switch points to the right: the axis stops feeding and the spindle stops. In the process of processing when the switch is rotated in the middle and right, the axis or spindle is only in a suspended state, this.

If the switch is returned (I. e. in the left position), the spindle program will continue to be started for processing. If the "reset" key is pressed at this time, the automatic machining operation state will be exited, and the current line refers to the 1 line of the machining program.

In the manual state, the start and stop of the spindle is not controlled by the intervention switch and is controlled by the button.

(2)G00 Quick Adjustment: Each time the "Quick ↑" key is pressed once, the feed speed magnification adjustment will be increased by 25%; Each time the "Quick ↓" key is pressed once, the feed speed magnification adjustment will be reduced by 25%. The rapid magnification can be adjusted arbitrarily according to different situations during processing.

(3) Feed speed trimming: when the comprehensive parameter 190 number is set to 0, the feed speed magnification trimming will be increased by 10% every time the "feed ↑" key is pressed; Each time the "feed↓" key is pressed, the feed speed magnification trimming will be reduced by 10%. When the integrated parameter 190 number is set to 1, the feed speed will be adjusted by the external band switch. The feed speed is adjusted to 0%-150 16 gears, and the processing speed can be adjusted arbitrarily according to different situations during the processing.

(4) Spindle speed adjustment: when the comprehensive parameter No. 189 is set to 0, the spindle speed ratio adjustment will be increased by 10% every time the "spindle ↑" key is pressed; Each time the "spindle↓" key is pressed, the spindle speed ratio adjustment will be reduced by 10%. When the integrated parameter 189 number is set to 1, the spindle speed will be adjusted through the external band switch. The spindle speed is adjusted to 0%-150 16 gears, and the spindle speed can be adjusted arbitrarily according to different situations during the processing.

(5) Stop during processing: During continuous processing, if you press the "Single Segment" key, the current segment program line will stop running after execution and wait for operation.

(6) Processing pause: The processing program can be paused by placing the three-stage switch in the middle, right, or pressing the "manual" key or turning on the external HALT stop signal. After pausing, set the intervention switch to the left or press the "Run" button or the "Start" button, the program will continue to run; if the "Reset" button is pressed at this time, the automatic machining operation state will be exited, and the current line points to the 1 line of the machining program.

(7) Feed maintenance: when the processing is suspended (external HALT, press the "manual" key or the "single segment" key), press the "manual" key to automatically feed maintenance, coordinate adjustment and other work can be carried out, then press the "automatic" key, and then press the "start" key to automatically run to the pause point and then continue to run.

(8) Exit processing: press "RESET" key during operation or "pause" or "feed hold.

4.5.6 MDI operation mode

Press the "MDI" key in the manual adjustment or automatic coordinate state to enter the MDI execution mode. "MDI" mode is to enter a program to execute a program, press the "return" key to abandon the exit during the input process, and press the "start" key to immediately execute the current input program segment.

4.5.7 CNC Machining Function

In this system, the user storage space is 100Mbit. When the processing program is greater than 100M or the remaining storage space is less than the program size, DNC function processing is required. CNC function in this system can be achieved through the USB interface.

USB-CNC Description:

The USB-DNC is implemented through a USB flash drive, that is, the USB flash drive is externally hung on the system, and the program in the USB flash drive is selected for execution.

Press the "Open USB" key under the program interface to open the USB flash drive. After selecting the corresponding program, press the "Run Program" key and press the "Auto" key to switch to the automatic state. At this time, press the "Run" key to run the program.

To stop the operation of the system during online machining, rotate the "three-position pause switch" to the center or the right end, and cancel the "stop" key or "RESET" key of online machining state.

Note: 1. Please do not pull out the USB flash drive during USB-CNC, let alone remove the USB flash drive, which will cause processing failure.
2. After the USB-CNC is completed, return to the system program interface from the USB interface.

4.6 safety operation and alarm prompt

4.6.1 Emergency stop

When an emergency accident occurs, press the "emergency stop" button, the system will stop all actions of the machine tool, and the system reset screen will display "emergency stop". Wait for the button to lift. The output state M67 outputs a valid signal.

If the "emergency stop" button is pressed during the machining process or during the movement of the machine tool, the system coordinates and the machine tool position may change. Before machining, the system coordinate position needs to be re-determined. It is better to perform the zero point operation of the machine tool to make the display coordinates consistent with the machine tool position.

The system is externally connected with the emergency stop ESP button, and the button is shielded, normally open or normally closed through the comprehensive parameter 188 number setting.

4.6.2 System Reset

At any time when the system is running, you can press the "RESET" key to stop the current operation, especially when you press the key manually and automatically to stop all machine actions such as worktable, spindle, tool rest, etc., but the coordinates will not be lost and there is no need to return to zero.

4.6.3 Machine tool alarm

When an alarm occurs on the machine tool, the screen will prompt an error message and flash in a reverse white manner. The program will stop running and the coordinate movement will be stopped. At this time, the cause of the fault should be checked and the fault can be eliminated before operation.

4.7 parameter operation

In the program stop state, press the "parameter" key to enter the parameter setting state. In this system, the parameters are divided into six categories: "basic parameters", "transmission parameters", "comprehensive parameters", "tool parameters", "screw parameters" and "other settings.

After entering the parameter interface, press the 8 keys directly below the display screen to enter the corresponding interface, press the "↑" and "↓" keys to select the parameter number, press the "Confirm" key to pop up a dialog box to input data, and then "Confirm" key to modify the parameters successfully after the input is completed.

Specific parameters are described as follows:

Basic parameters			
Parameter No.	Parameter Description	Unit	Factory setting value
70	Chamfer width for thread cutting (G92,G76)	0.1*pitch	0
136	Depth of cut for multiple fixed cycles (G71,G72)	um	0
137	Regression for multiple fixed cycles (G71,G72)	um	0
138	Multiple fixed cycles (G73) Total cut on X-axis	um	0
139	Total cut in Z-axis for multiple fixed cycles (G73)	um	0
140	Number of cycle cuts for multiple fixed cycles (G73)		0
141	Regression for multiple fixed cycles (G74,G75)	um	0
142	Number of repetitions of multiple fixed cycle (G76) finishing		0
143	Tip angles 0, 29, 30, 55, 60, 80 for multiple cycles	Degree	60
144	Minimum depth of cut in multiple fixed cycles (G76)	um	0
145	Finishing allowance in multiple fixed cycles (G76)	um	0
152	M Code Execution Duration	ms	0
153	S Code Execution Duration	ms	0

160	G50 Minimum speed limit	R/MIN	0
Transmission parameters			
24	Spindle jog time	ms	100
34	Pulse frequency for backlash compensation (common to each axis)		5
35	Manual mechanical return to zero speed (shared by all axes)		3000
38	M600 encoder Z-pulse speed		200
58	X-axis coordinate instruction multiplication ratio		1
59	Y-axis coordinate instruction multiplication ratio		1
60	Z-axis coordinate instruction multiplication ratio		1
61	A- axis coordinate instruction multiplication ratio		1
62	B- axis coordinate instruction multiplication ratio		1
63	C- axis coordinate instruction multiplication ratio		1
64	X-axis command frequency division factor		1
65	Y-axis command frequency division factor		1
66	Z-axis command frequency division factor		1
67	A- axis command frequency division factor		1
68	B- axis command frequency division factor		1
69	C- axis command frequency division factor		1
71	Workpiece coordinate attributing (D5-C,D4-B,D3-A,D2-Z,D1-Y,D0-X) 1-Absolute Coordinate Circumference Absolute, 0-Not Absolute		00000000
72	X-axis fast rate	MM/MIN	7000
73	Y axis fast rate	MM/MIN	7000
74	Z-axis fast rate	MM/MIN	7000
75	A- axis fast rate	MM/MIN	7000
76	B- axis fast rate	MM/MIN	7000
77	C- axis fast rate	MM/MIN	7000
78	X coordinate linear acceleration and deceleration time constant (for fast movement)	MS	200
79	Y coordinate linear acceleration and deceleration time constant (for fast movement)	MS	200
82	Z-coordinate linear acceleration and deceleration time constant (for fast movement)	MS	200
81	A coordinate linear acceleration and deceleration time constant (for fast movement)	MS	200
82	B coordinate linear acceleration and deceleration time constant (for fast movement)	MS	200
83	C coordinate linear acceleration and deceleration time constant (for fast movement)	MS	200
90	thread cutting acceleration and deceleration constant	MS	20
91	Cutting feed upper limit speed	MM/MIN	4000
96	Acceleration and Deceleration Time Constants in Hand-vein Cutting Feed	MS	200
97	Linear acceleration and deceleration time constant in cutting feed	MS	200
98	Minimum speed of fast moving magnification (F0)	MM/MIN	100
99	Returns the low,(FL) speed of the reference point	MM/MIN	100
100	X-axis clearance compensation	UM	0
101	Y-axis clearance compensation	UM	0
102	Z-axis clearance compensation	UM	0
103	A- axis clearance compensation	UM	0
104	B- axis clearance compensation	UM	0
105	C- axis clearance compensation	UM	0
107	Spindle speed of gear 1 when spindle command is 10V	R/MIN	3000
108	Spindle speed of gear 2 when spindle command is 10V	R/MIN	3000
109	Spindle speed of gear 3 when spindle command is 10V	R/MIN	3000
110	Spindle speed of gear 4 when spindle command is 10V	R/MIN	3000
111	When the spindle command is 10V, the second spindle speed	R/MIN	3000
116	Spindle speed limit under constant linear speed control	R/MIN	100
117	Maximum inching speed	MM/MIN	6000
118	X-axis negative travel limit	UM	-199999998

119	X-axis forward travel limit	UM	199999998
120	Y-axis negative travel limit	UM	-99999999
121	Y-axis forward travel limit	UM	99999999
122	Z-axis negative travel limit	UM	-99999999
123	Z-axis forward travel limit	UM	99999999
124	A- axis negative travel limit	UM	-99999999
125	A- axis forward travel limit	UM	99999999
126	B- axis negative travel limit	UM	-99999999
127	B- axis forward travel limit	UM	99999999
128	C- axis negative travel limit	UM	-99999999
129	C- axis forward travel limit	UM	99999999
130	Set the X value in the automatic coordinate system after returning to zero	UM	0
131	Set the Y value of the automatic coordinate system after returning to zero	UM	0
132	Set the Z value of the automatic coordinate system after returning to zero	UM	0
133	Set A value of automatic coordinate system after returning to zero	UM	0
134	Set B value of automatic coordinate system after returning to zero	UM	0
135	Set C value of automatic coordinate system after returning to zero	UM	0
146	Number of spindle encoder lines		1024
147	Shift time 1		0
148	Shift time 2		0
149	Spindle speed when shifting		0
161	Pitch compensation: 1-valid, 0-invalid		0
162	Pitch error compensation rate (common for each axis)		1
163	X-axis pitch error compensation origin 0-255		0
164	Y axis pitch error compensation origin 0-255		0
165	Z-axis pitch error compensation origin 0-255		0
166	A axis pitch error compensation origin 0-255		0
167	B- axis pitch error compensation origin 0-255		0
168	C- axis pitch error compensation origin 0-255		0
169	X-axis pitch error compensation clearance	UM	0
170	Y axis pitch error compensation clearance	UM	0
171	Z-axis pitch error compensation clearance	UM	0
172	A axis pitch error compensation clearance	UM	0
173	B- axis pitch error compensation clearance	UM	0
174	C- axis pitch error compensation clearance	UM	0
175	Maximum cutting feed rate per feed	UM/R	2000
comprehensive parameter			
0	Spindle control mode: 0-inverter control or analog control, 1-others		0
1	Handwheel mode: 1-handheld unit, 0-standard mode		0
2	X-axis programming: 1-diameter programming, 0-radius programming		1
3	Chuck form: 1-card, 0-card		0
4	Electric tool holder: 0-not installed, 1-installed,2-Other methods,3-Liuxin Knife Pagoda		0
5	Lathe form: 0-sleeping car, 1-inclined car, 2-vertical car		0
6	Whether the fast motion command is valid during idle operation: 0-invalid, 1-valid		1
7	Return to zero level (D5-C,D4-B,D3-A,D2-Z,D1-Y,D0-X) 0-low level, 1-high level		00000000
8	External chuck control M16:1-valid, 0-invalid		1
9	External tailstock control M14:1-valid, 0-invalid		1
11	Chuck control: 1-valid, 0-invalid		1
13	Spindle at S code: 1-automatic shift, 0-manual shift		0

14	M30 cursor return to the beginning of the way: 0-cycle start return again, 1-return immediately		1
15	Formatting system is valid: 0-valid, 1-invalid		0
16	Does absolute zero programming (G50) set relative coordinates: 0-do not set, 1-set		0
17	Pulse output form: 0-pulse + direction, 1-double pulse		1
19	When non-standard MST code is programmed: 0-generate alarm, 1-do not alarm		0
20	Back to zero direction:(D5-C,D4-B,D3-A,D2-Z,D1-Y,D0-X) 0-negative, 1-positive		00111111
21	Chuck loose signal time (ms)		0
22	Tailstock backoff signal time (ms)		0
23	Zero back detection Z pulse: 0-no detection, 1-detection		0
25	Motor direction:(D5-C,D4-B,D3-A,D2-Z,D1-Y,D0-X) 0-same direction, 1-inverse		00000000
26	Shaft installation:(D5-C,D4-B,D3-A,D2-Z,D1-Y,D0-X) 0-do not install, 1-install		00000101
27	Multiplexing K1,K2:0-Output Y24Y25, 1-K1K2 as C- axis positive and negative keys respectively		0
28	M601 servo spindle positioning function: 0- invalid 1- valid		0
29	Machine Door Switch: 0-Invalid, 1-Valid		0
30	Machine door switch level: 0-low level, 1-high level		0
31	Drive alarm levels (D3-ALM3,D2-ALM2,D1-ALM1,D0-ALM): 0-low level, 1-high level		00001111
33	M00 Allowable to change the auxiliary status (M03,M04,M05,M10,M11,M32,M33) when the program is suspended: 0-not allowed, 1-allowed		0
36	M99 program cycle number: -1 is the infinite cycle, the positive number is the number of cycles, please change and press the reset key to take effect		-1
37	maximum number of workpieces to be run (0: invalid)		0
38	M600speed		200
40	Tool holder in place level: 0-active low level, 1-active high level		0
41	Tool holder locking level: 0-low level effective, 1-high level effective		0
42	Manual direction keys reverse (D5-C, D4-B, D3-A, D2-Z, D1-Y, D0-X) 0- not reverse, 1-reverse)		0
43	T1-T4 multiplexing (0: knife holder, 1: external input signal switch)		0
44	T5-T7 multiplexing (0: knife holder, 1: external input signal switch)		0
45	M65 M67 M69 multiplexing: 0-machine tool three color light output 1-default output port		0
46	ALM alarm when off M5 spindle: 1-off, 0-off		0
47	Manual quick before zero return: 1-valid, 0-invalid		0
48	Spindle operation and chuck interlock: 0-no interlock, 1-interlock		0
49	Spindle running and tailstock interlock: 0-no interlock, 1-interlock		0
51	Single program segment when executing macro instruction: 1-valid, 0-invalid		0
52	Mechanical zero point (D5-C,D4-B,D3-A,D2-Z,D1-Y,D0-X): 1-Yes, 0-No		00000000
115	Automatically insert program sequence number when editing: 1-insert, 0-do not insert		0
158	Spindle command stop to spindle brake output time	MS	0
159	Spindle brake output time	MS	0
177	Spindle automatic shift: 1-valid, 0-invalid		0
182	Tailstock control function: 1-valid, 0-invalid		1
183	Chuck control mode: 1-double signal, 0 single signal		1
184	Tailstock control mode: 1-double signal, 0 single signal		1
185	External loop start (ST) signal: 0-Masked, 1-Unmasked		1
186	External pause (SP) signal: 0-shielded, 1-unshielded		1

187	Soft Limits (D5-C,D4-B,D3-A,D2-Z,D1-Y,D0-X): 0-do not check, 1-check		00000000
188	Emergency stop (ESP) signal: 0-shielded, 1-unshielded		1
189	Spindle magnification switch: 1-valid, 0-invalid		0
190	Feed rate switch: 1-valid, 0-invalid		0
191	Three-segment switch: 1-valid, 0-invalid		1
192	Lubrication opening time:	MS	0
193	Lubrication gap time:	MS	0
194	Absolute encoders (D5-C,D4-B,D3-A,D2-Z,D1-Y,D0-X): 0-No, 1-Yes		00000000
195	Absolute encoder multi-turn address		92
196	Absolute encoder single turn high address		91
197	Absolute encoder single turn low address		90
198	Shaft absolute encoder lines		131072
199	Corresponding coordinates of each circle of X-axis absolute encoder	NM	10000000
200	Y-axis absolute encoder corresponding coordinates for each turn	NM	10000000
201	Coordinates corresponding to each circle of Z-axis absolute encoder	NM	10000000
202	Coordinates corresponding to each turn of A- axis absolute encoder	NM	10000000
203	B- axis absolute encoder corresponding to the coordinates of each turn	NM	10000000
204	Coordinates corresponding to each turn of C- axis absolute encoder	NM	10000000
205	X-axis absolute encoder position offset	UM	0
206	Y-axis absolute encoder position offset	UM	0
207	Z-axis absolute encoder position offset	UM	0
208	A- axis absolute encoder position offset	UM	0
209	B- axis absolute encoder position offset	UM	0
210	C- axis absolute encoder position offset	UM	0
211	During emergency stop (D7-M65,D6-M7118,D5-M7117,D4-M7116,D3-M59,D2-M63,D1-M61,D0-M75): 0-Hold, 1-Close		00000000
212	During emergency stop (D4-M08, D3-M79, D2-M32, D1-M69, D0-M67): 0-Hold, 1-Close		00000000
213	When resetting (D7-M65, D6-M7118,D5-M7117,D4-M7116,D3-M59,D2-M63,D1-M61, D0-M03/04): 0-Hold, 1-Close		00000000
214	When resetting (D5-M75, D4-M08, D3-M79, D2-M32, D1-M69, D0-M67): 0-Hold, 1-Close		00000000
Tool parameters			
150	Upper limit of time required to move one knife position	MS	6000
151	The upper limit of time for the first 1 to change to the last knife	MS	6000
154	Tool change T1 time	MS	100
155	Alarm time of * TCP not received	MS	0
156	Total knife position quantity		4
157	Tool holder reverse lock	MS	1200
Screw complement parameter			
Serial Number	X Axis	Z-axis	Y axis
0-255	0	0	0
Other Settings			
0	Backup system parameters to the current system		
1	Backup system parameters to U disk		
2	Restore system parameters from the current system		
3	Recover System Parameters from U Disk		
4	Restore factory default system parameters		
5	Back up the screw patch parameters to the current system		
6	Back up the screw patch parameters to the U disk		
7	Recover Screw Complement Parameters from Current System		
8	Recover Screw Complement Parameters from U Disk		

9	Empty screw fill parameter		
10	Set system time		
11	Set system date		
12	System internal parameters:		

Alarm number	Alarm reason
000	Please cut off the power
001	File open failed
002	Input data not recognized
003	Data entered in excess of the allowed number of bits
004	There are only numbers or symbols at the beginning of the program segment and no address words.
005	No data after the address, followed by the next address ", or EOB code.",
006	Illegal address word input ", unrecognizable characters appear in the program
007	The decimal point input error, (input at the address that does not allow decimal point input, decimal point or input more than two decimal points)
008	File creation was unsuccessful.
009	Illegal word address character entered
010	Instruction of illegal G-code
011	The feed speed is not specified in the cutting feed or the feed speed command is inappropriate.
012	The coordinates of the target point exceed the maximum programmed size
013	Maximum speed of rapid feed is less than or equal to zero
014	Coordinate out of bounds during circular interpolation
015	Current speed at start of interpolation is not zero
016	The spindle command speed in thread interpolation is too small, or the constant speed limit command is not canceled.
017	divide by zero error
018	Thread interpolation lead less than or equal to zero
019	It is not a thread interpolation instruction that calls thread interpolation.
020	The line interpolation is not called a line interpolation instruction.
021	Radius compensation R value is zero
023	In circular interpolation specified with radius R
024	In circular interpolation, the radius of circular arc is greater than the value range
025	It is not an arc interpolation instruction that calls arc interpolation.
026	The data given in circular interpolation cannot form an arc
027	In circular interpolation, R and (K,I) are specified at the same time.
028	R and (K,I) are all zero in circular interpolation
029	The bias value of the instruction with T code is too large
030	The number of the knife deviation exceeds the value range.
031	The file cannot be searched in automatic mode.
032	Arc end not on arc
033	No intersection in nose radius compensation
034	When the tool radius compensation is started or the tool radius compensation is canceled, the moving program segment of "is an arc command
035	There is a no-move command when the knife is in progress, After a row, another no-move command appears.
036	The tool radius compensation start segment has a tip radius compensation number of 0
037	The next 1 segment of the compensation start segment changes the direction of the tool compensation.
038	The center of the arc coincides with the start point or end point
041	In the process of tool compensation, tool path interference
042	When the tool radius is compensated, the arc command repeats the specified tip ", radius compensation G code",
043	Entered data is out of allowed range"
050	S-type acceleration time is less than zero
051	S-type deceleration time is less than zero
052	Linear acceleration and deceleration time less than zero
053	G00 Acceleration and deceleration The acceleration and deceleration time of the X axis is less than zero
054	G00 Acceleration and deceleration The acceleration and deceleration time of the Z axis is less than zero

062	(1) The depth of cut in G71 or G72 is zero or negative. ", (2) The number of repetitions of G73 is zero or negative. ", (3) D-i or D-k instructions in G74 or G75 are negative. ", (4) Although the D-i or D-k in G74 or G75 is not zero, ", the address U or W is specified as zero or negative.", (5) Although the retreating direction of G74 or G75 is specified, ", the D-d is negative.
063	In G70,G71,G72 or G73 instructions, the sequence number specified by address P cannot be retrieved
064	When the G code is G32,G33,G90--G94,G70--G76, the knife repair is not canceled,
065	(1) In G71,G72 or G73 instructions, the program segment with the sequence number specified by address P "does not instruct G00 or G01.", (2) in the G71 or G72 instruction, in the program segment of the sequence number specified by address p, "address Z(W)(G71) or X(U)(G72) is instructed
066	In the G70,G71 or G72 instruction ", in the two program segments specified by address P or Q", Instruction cannot be used G-code
074	The program number is not within the specified range.
076	In the program segment of M98, P is not specified
077	Too many nested subroutine calls
078	The program number or sequence number specified by P was not found in the M98 or M99 program segment.
081	The automatic tool compensation command was commanded before the T code was commanded. ",
090	When returning to the reference point, the start point is too close to the reference point or the speed is too slow, and the return to the reference point cannot be executed normally
100	Parameter switch is ON
101	In the program editing, when the memory is rewritten, the power is cut off, and the alarm will be automatically canceled after shutdown. (Note whether the parameters, procedures, etc. have changed)
111	The operation result of the macro program command exceeds the allowable range.
112	The number of data in the parameter file is greater than the total number of parameters
113	Parameter file contains non-numeric characters
114	In the program segment of G65, the undefined H code is instructed
115	Illegal variable number specified
125	In the program segment of G65, the address that cannot be used is instructed.
128	In the transfer instruction, the sequence number of the transfer address is not within the ", allowed range. Or the sequence number to be transferred was not found.
129	Two identical function words cannot appear in the same program segment
130	An address value is out of range
131	G00 group and G01 group cannot appear at the same time
132	Coordinate values X, U, Y, V,Z and W cannot appear at the same time
134	G32,I, K appear at the same time
135	G04, neither P nor X
136	G50 coordinate initial value error
138	N must be incremented
139	G33 Headless Value
140	There should be a space between addresses
142	Program ended illegally without M30
143	G32,F, I initial value not assigned
144	G70, G71, G73, G74, G75, G76, address word value error
145	Variable subscript out of range
146	Address O and N cannot refer to variables
147	P cannot be a constant in G65
148	divide by zero error
149	Tool radius compensation error not canceled when entering or exiting subroutine
150	The rough machining trajectory specified in the G71 and G72 instructions ", the change of boundary coordinate values is not monotonic.
151	In G71 and G72 instructions, the positive and negative values of u and w "cannot match the trajectory shape",
152	When machining the cone cutting cycle, it exceeds G90,G92,G94 and can handle the work cycle.

153	No non-modal instruction in G90, G92, G94 ",F and I are assigned or F or I specified is inappropriate
154	Simultaneous assignment of F and I during thread insertion
156	G71,G72,G73Pno must be located next to G71,G72,G73
157	The sequence number specified by address Q in G70,G71,G72 or G73 instruction cannot be retrieved
158	G70,G71,G72, or G73 specifies a P value greater than or equal to the Q value
159	In the G71 or G72 instruction, in the program segment of the sequence number specified by the address P, the address Z(W)(at G71) or X(U)(at G72) is instructed
162	In the segment of the program that calls the subroutine, P is a non-integer value, or a negative value.
163	The number of subroutine calls is greater than the specified number.
165	Illegal Program Segment (G90-G94,G70-G76)
166	G70,G71,G72,G73 assigned Pno and Qno values, out of range or not integer
201	At least one shaft is out of the forward travel limit,
202	At least one axis exceeds the negative travel limit
203	The spindle is running and the chuck cannot be operated.
204	Spindle running can not operate the tailstock
205	The spindle cannot be opened without clamping the chuck.
206	The tailstock is not clamped and the spindle cannot be opened.
207	Overtravel or emergency stop switch alarm
210	Emergency stop
211	Servo drive alarm
212	Spindle Alarm ALM1
213	Machine tool alarm ALM2
214	Machine tool alarm ALM3 (bus communication alarm)
222	Z-axis drive alarm
300	F code is wrong. Negative speed values programmed
301	M code error, illegal M code programmed into the program
302	S code is wrong. Illegal S-code programmed into the program
303	T code wrong. Illegal T-code programmed into the program
305	Tool change time is too long. Starting from the tool holder to rotate forward, the specified tool position reaches the signal after Ta time
306	When it is still not received, an alarm is generated and the system parameter is loaded incorrectly.
307	Tool compensation parameter loading error
308	Screw fill parameter loading error
309	System parameter saving error
310	Parameter saving error
311	Error saving screw parameters
400	Spindle speed is zero
500	NvRam error
501	The spindle speed is too fast or the thread is too short in thread interpolation
502	Program length out of system range
601	At least one shaft is out of forward soft limit
602	At least one axis exceeds the negative soft limit
610	X-axis absolute value read error
611	Y-axis absolute value read error
612	Z-axis absolute value reading error
613	A- axis absolute value reading error
614	B- axis absolute value reading error
615	C- axis absolute value reading error
616	No jump number found
700	System internal error, please contact the manufacturer
800	maximum number of workpieces exceeds

4.8 edit operation

Under any menu, press the "Edit" key to enter the programming state. The program management adopts the file management mode. The system can store 100M byte capacity programs, and the number of programs is 1000. Program editing using full-screen mode of operation.

On the left side of the program editing interface is the program directory area, which displays all the program names and folders in the memory, and the blue display is the current program or folder. You can select the program arbitrarily through the “↑↓”, upper page, lower page" keys, and display it in yellow. Press the "OK" key to enter the corresponding operation state.

4.8.1 New File

After selecting the "New File" function, the system will pop up a dialog box to enter the program name. If you enter a file name that already exists in the system, you will be prompted that the file already exists. If the entered file name does not exist, a new file will be created. Program name can be numbers, letters, Chinese or mixed, length of 100.

The system does not allow the name of the program name, create a new program name or select a program name and press the "OK" key to enter the program editing interface.

Input method for file name and program content:

In the editing state, the upper left corner of the screen prompts the edited program name; the left side is the program content, and the right side is the machine tool status information area. In the editing state, the following operations can be performed:

(1) Current cursor position: Press the ↑ ↓ ← → key to move the cursor to any position in the program content. Press the Previous Page key to move the cursor to the previous page. Press the Next Page key to move the cursor to the next page.

(2) Character modification: delete the character to be changed at the cursor, and then directly enter the new character.

(3) Character insertion: Enter a new character directly at the cursor. When the input is a letter, a space is automatically generated in front of the letter. If you want to enter a space, enter a letter first and then delete the letter.

(4) character delete: directly press the "delete" key at the cursor.

(5) Insert between lines: Press the "OK" key directly. If the cursor is inserted at the beginning of the line, a line will be inserted before the current line, otherwise a line will be inserted after the current line.

4.8.2 Copy

In the editing main interface, press the "↑" and "↓" keys to select the program name to be copied, press the "Copy" key, then press the "Paste" key, press the "Return" key, and enter the new program name in the system pop-up dialog box, that is, copy a program with different program name and exactly the same content, so as to modify, back up and rename.

4.8.3 Delete

In the main interface of the program, press the "↑" "↓" keys to select the program or folder to be deleted, and press the "Delete" key to delete the program. Be careful when deleting, the program cannot be restored.

4.8.4 Renaming

In the main interface of the program, press the "↑" and "↓" keys to select the program to be renamed, then press the "Rename" key to enter a new file name in the system pop-up dialog box, and then rename the current program to the newly entered program name.

4.8.5 Folder Management

In this system, files can be established. Press the "**New Folder**" key in the main interface of the program to enter the file name and then press the "Confirm" key to establish a folder, that is, to establish a folder, and the "folder" after its size name is marked. Move the cursor to the file name and press the "confirm" key to open the folder and create new files and folders in it. Press "superior directory" for the next folder to the previous folder. To delete a folder, you must delete the contents of this folder before you can delete the folder.

4.8.6 Selection of automatic operation program

In the program main interface, press the "↑" and "↓" keys to select the program to be executed and press the "Run Program" key. The system will automatically select the program and switch to the main interface (the main interface before entering the program).

4.8.7 U disk management

In this system, the program exchange with the C disk system and the upgrade and backup of the system software can be

carried out through the USB interface.
Note: When using a USB flash drive, the folder name cannot have a space symbol.
After inserting the USB flash drive into the USB interface, select "USB disk" under the main interface of the program to enter the USB flash drive management interface. Press the "C Disk" key again to return to the system interface.

1. Copy the program files in the processing program management USB disk into the system:

After inserting the USB flash drive into the system, select "USB disk" in the editing main interface, press the "confirm" key to enter the USB disk directory, press the "↑" and "↓" keys to move the cursor to select the program file to be copied into the system, press the "Copy" key at this time, press the "↑" and "↓" keys to move the cursor to select "C disk", press the "confirm" and then press the "Paste" key, if the current file on the U disk is copied into the system, the system will automatically switch to the main program interface. If the typed program file exists, the information of whether to overwrite the original program file or folder will pop up.

2. Copy the program files in the system to USB flash drive:

After inserting the USB flash drive into the system, in the editing main interface, press the "↑" and "↓" keys to move the cursor to select the program file to be copied into the system. At this time, press the "Copy" key, then press the "↑" and "↓" keys to move the cursor to select "USB disk", press "Confirm" and then press the "Paste" key to copy the current file on the C disk into the USB disk, if the typed program file already exists, a message will pop up whether to overwrite the original program file.

3. Upgrade management of system software

In this system, the system software can be transmitted through the USB interface, so as to achieve the purpose of upgrading. Its specific operation method is as follows:

Import (upgrade, update) system software with USB flash drive:

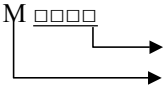
After inserting the USB flash drive into the USB interface of the system, press the "↑" and "↓" keys to move the cursor to select "USB disk" and press "confirm" on the panel to select the file to be upgraded. then press the "u" key and the system will prompt "new system software", press the "confirm" key, and then the system will prompt "update is complete, restart the system!" The system automatically reloads the new software to upgrade the software.

Note: The parameters to be imported must be set up in a separate folder in the computer, and then copied to the root directory of the USB flash drive for import to prevent damage to the system file system by operation error.

M-Code (Accessibility)

The M instruction consists of an instruction address M followed by 1 to 2 digits or 4 digits, and is used to control the process of program execution or output M code to PLC.

M □□□□



Instruction value (00~99, 9000~9999, leading zero can be omitted)
instruction address

M 98, M 99, M9000 to M9999 are independently processed by NC.

M02 and M30 have been defined by NC as program end instructions, and M code is also output to PLC, which can be used by PLC program for input

Output control (off spindle, off cooling, etc.).

M 98, M 99, and M9000 to M9999 are program call instructions, and M02 and M30 are program end codes.

There can only be one M code instruction in a program segment. When two or more M codes appear in the program segment,

CNC alarm. Table 2-1 List of Process M Instructions for Control Program Execution

Instruction	Function
M02	End of program running
M30	End of program running
M20	Number of processed pieces plus 1
M97	Procedural Transfer
M98	subroutine call
M99	Return from subroutine; if M is used at the end of the main routine (I. e., when The former program is not called by other programs), and the program is executed repeatedly.
M9000 -M9999	Call macro program (program number greater than 9000)

End of program M02

Command format: M02 or M2

Instruction function: in the automatic mode, the M02 instruction is executed. After the other instructions of the current program segment are executed, the automatic operation ends, The cursor stays in the program segment where the M02 instruction is located and does not return to the beginning of the program. To execute the program again, you must return the cursor to the beginning of the program. In addition to the functions of the NC processing described above, the current output state of the CNC remains unchanged after the M02 command is executed.

End of program run M30

Instruction format: M30

Instruction function: In the automatic mode, the M30 instruction is executed. After the other instructions of the current program segment are executed, the automatic operation ends, Add 1 to the number of processed pieces, cancel the tool tip radius compensation, and return the cursor to the beginning of the program (whether to return to the beginning of the program is determined by the parameter).
When the number of the system integrated parameter 14 is set to 0, the cursor does not return to the beginning of the program; when the system integrated parameter 14 number is set to 1, After the program is executed, the cursor immediately returns to

the beginning of the program. After executing the M30 instruction, turn off the M03 or M04, M08 signal output, and output the M05 signal at the same time.

Skip function:

Two ways to program arbitrary segment start:

1. format of instruction:/M97 P; program transfer

Instruction function: P-the program segment number transferred to, the leading zero of the program segment number can be omitted, and the segment number specified by P cannot be the program segment of M97 itself. Under the manual or automatic interface, you can press the/key to turn the jump on or off, and make the jump function choose between jump and not pick without deleting this program segment.

2. In the automatic state, enter the letter "N" and then enter the mark segment number, then press the "Enter" key, and then press the "Start" key

Note: To start any section of the above two methods, the tool number to be run must be manually retrieved first, and the spindle, water pump and chuck must be manually run,
Run the skip function again.

subroutine call M98

Command format: M98 P ○ ○ ○ ○ □ □ □ □

the number of the called subroutine (0000~9999). When the number of calls is not entered, the leading 0 of the subroutine number can be omitted;

When using times, the subroutine number must be 4 digits;

Number of calls (1-9999). When one call is made, no input is required.

Instruction function: in the automatic mode, the execution of M98 instructions, the current program segment of other instructions after the completion of the CNC to call Execute the subroutine specified by P, the subroutine can be executed up to 9999 times. The M98 command does not work under MDI.

Return from subroutine M99

command format: M99 P○○○○

Program segment number to be executed when returning to the main program(0000~9999), leading 0 can be omitted.

Instruction function: (in subroutine) After the execution of other instructions in the current program segment is completed, return to the program segment specified by P in the main program to continue execution, When P is not entered, the 1 program segment of the M 98 instruction that calls the current subroutine in the main program is returned to continue execution. If M is used for the end of the main program (I. e., the current program is not called by another program), the current program will execute repeatedly. The M99 instruction is not valid for running under MDI. The current program will be executed repeatedly, and the number of program cycles is determined by the comprehensive parameter 36.

Example: Figure 2-1 shows the execution path of the calling subroutine (M99 has a P instruction word).
Figure 2-2 shows the execution path of the calling subroutine (no P instruction word in M99)

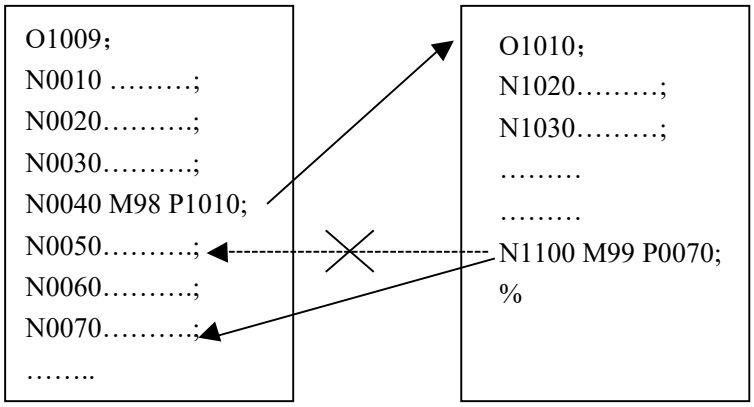


Figure 2-1

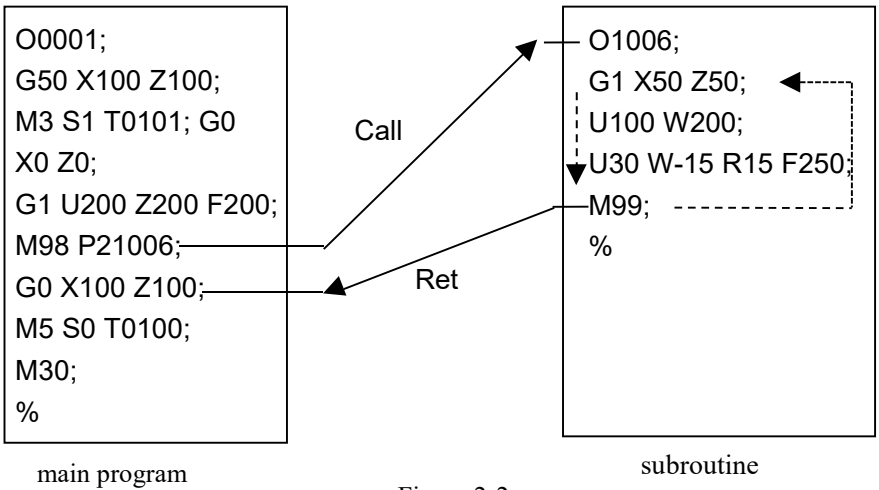

O0001;
G50 X100 Z100;
M3 S1 T0101; G0
X0 Z0;
G1 U200 Z200 F200;
M98 P21006;
G0 X100 Z100;
M5 S0 T0100;
M30;
%
Call
Ret
O1006;
G1 X50 Z50;
U100 W200;
U30 W-15 R15 F250;
M99;
%
main program
subroutine

Figure 2-2

This system is capable of calling quadruple subroutines, I .e., subroutines within which other subroutines can be called (as shown in Figure 2-3.

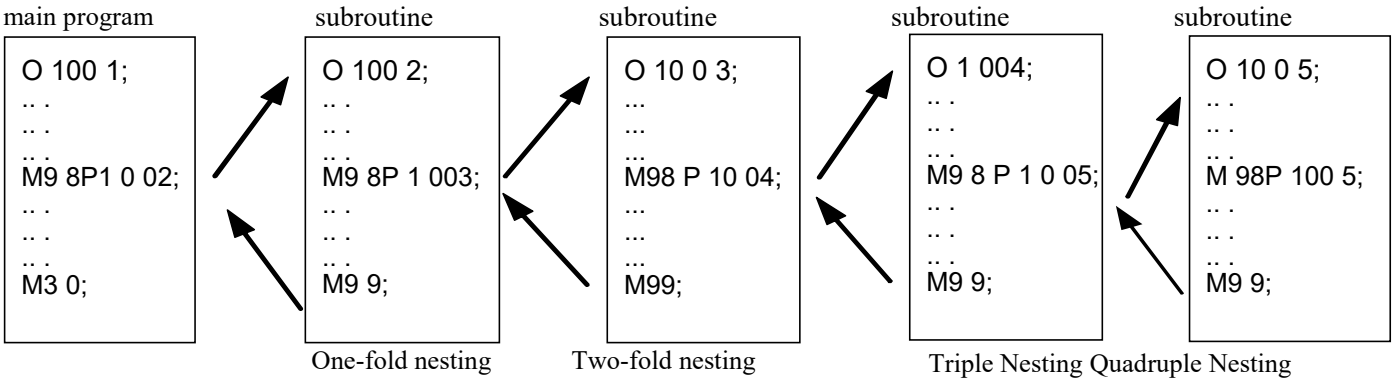


Figure 2-3 Nested subroutines

Add 1:M20 to the calculation of the number of processed pieces

Instruction format: M20

Instruction function: the program runs M20, and the number of processed pieces is increased by 1.

M instruction defined

Instruction	Function	Remarks
M00	Program suspension	
M03	spindle forward rotation	Function Interlock, Status Hold
M04	Spindle reversal	
M05	Spindle stop	
M08	Coolant on	Function Interlock, Status Hold
M09	Coolant Off	
M79	Tailstock entry	Function Interlock, Status Hold
M78	Tailstock retreat	
M11	chuck clamping	Function Interlock, Status Hold
M10	chuck loosening	
M32	Lubrication ON	Function Interlock, Status Hold
M33	Lubrication Off	
M75	Output On	Function Interlock, Status Hold
M74	Output Off	
M61	Output On	Function Interlock, Status Hold
M60	Output Off	
M63	Output On	Function Interlock, Status Hold
M62	Output Off	
M59	Output On	Function Interlock, Status Hold
M58	Output Off	
M41, M42, M43, M44	Automatic spindle shift	Function Interlock, Status Hold
M71xx	Output On	M71XX rear input signal bit number
M70xx	Output Off	M70XX Rear Input Signal
M61xx	Input Valid Signal	M61XX rear input signal bit number
M60xx	Input Invalid Signal	M60XX rear input signal bit number
M1800	Multi-channel communication waiting for instructions	

Program stop M00

Command format: M00 or M0

Instruction function: After executing the M00 instruction, the program stops running, and the word "pause" is displayed. After pressing the cycle start key, the program continues to run

Spindle forward and reverse stop control M03, M04, M05

Instruction format: M03 or M3;

M04 or M4;

M05 or M5;

Command function: M03: spindle forward rotation;

M04: Spindle reversal; M05: Spindle stops.

Cooling pump control M08, M09

Instruction format: M08 or M8;

M09 or M9;

Command function: M08: cooling pump on;

M09: Cooling pump off.

Tailstock control M79, M78

Instruction format: M79;

M78;

Command function: M79: tailstock entry;

M78/M73: tailstock back,

Chuck control M10, M11

Instruction format: M10;

M11;

Command function: M10: chuck clamping;

M11 : chuck release

Lubricating fluid control M32, M33

Instruction format: M32;

M33;

Command function: M32: lubrication pump on;

M33: Lubrication pump off.

Spindle automatic shift M41, M42, M43, M44

Instruction format: M4n;(n = 1, 2, 3, 4)

Command function: when executing M4n, the spindle shifts to the nth gear

Custom interface signal on output M71xx, custom interface signal off output M70xx

Instruction format: M71xx;

M70xx;

Command function: M71xx output is on, "xx" is the signal bit number, and the output status of each signal bit number "O" can be viewed on the diagnostic interface,

For example: M03, bit number O06, Input M7106 is equal to the functional meaning of M03.

M70xx output is off, "xx" is the signal bit number, and the output status of each signal bit number "O" can be checked on the diagnosis interface,

For example: M05, bit number O04, Input M7004 is equal to the function of M05.

Custom interface signal input valid M61xx, custom interface input invalid M60xx

Instruction format: M61xx;

M60xx;

Command function: M61xx input is valid for 0V, "xx" is the signal bit number, and the input status of each signal bit number "I" can be checked on the diagnostic interface,

For example: M12, the bit number is I37, and the input M6137 is equal to the functional meaning of M12.

M60xx input is invalid 0V, "xx" is the signal bit number, and the input status of each signal bit number "I" can be checked on the diagnosis interface,

For example: M13, the bit number is I37, and the input M6037 is equal to the functional meaning of M13.

Spindle function

S command is used to control the spindle speed, control the spindle speed in two ways:

Spindle speed switching value control mode: S□□□□(2-digit command value) command outputs switching value signal to the machine tool to realize spindle rotation. There are rapid changes.

Spindle speed analog voltage control mode: S□□□□(4-digit command value) specifies the spindle actual speed, NC output 0~10V.Analog voltage signal to the spindle servo device or inverter, to achieve the spindle speed stepless speed regulation.

Spindle speed switch control

When the comprehensive parameter 0 number is set to 1, the spindle speed is controlled by switching value. A program segment can only have one S instruction. If there are two or more S instructions, the last S instruction shall prevail.

Instruction format: S □ □

00~04 (leading zero can be omitted):1~4 gear spindle speed switching value control.

When the CNC is reset, the output states of S01, S02, S03 and S04 remain unchanged.

When CNC is powered on, S1 ~ S4 outputs are invalid. Execute any instruction in S01, S02, S03, S04, and the corresponding S signal output is valid

And hold, while canceling the output of the remaining 3 S signals. When executing the S00 command, cancel the output of S1 ~ S4, S1 ~ S4 at the same time only one effective.

Simulation Voltage Control of Spindle Speed

When the comprehensive parameter No. 0 is set x to 0, the spindle speed is controlled by analog voltage.

Instruction Format: S OOOO

0000~9999 (leading 0 can be omitted): spindle speed analog voltage control

Command function: set the speed of the spindle, CNC output ± 10V analog voltage to control the spindle servo or frequency converter, to achieve stepless speed change of the spindle,

S command value is not memorized when power is off, and is set to 0 when power is on.

When the spindle speed analog voltage control function is effective, there are two ways to input the spindle speed: S command to set the fixed speed of the spindle (rpm),

When the S command value does not change, the spindle speed is constant, which is called constant speed control (G97 mode);S command sets the tool relative to the outer circle of the workpiece

Tangent speed (m/min), called constant line speed control (G96 mode), constant line speed control mode, cutting feed spindle speed with

The absolute value of the X-axis absolute coordinate value of the trajectory changes.

CNC has the function of 4 gear spindle mechanical gear. when executing the s command, according to the maximum spindle speed of the current spindle gear (output analog electricity

Pressure is 10V) set value (corresponding to transmission parameter, 107-110 number) to calculate the analog voltage value corresponding to the given speed, and then output to the spindle

Servo or frequency converter, control the actual speed of the spindle is consistent with the required speed.

When the CNC is powered on, the theoretical analog voltage output is 0V. After the S command is executed, the output analog voltage value remains unchanged (unless it is in a constant line

speed control of the cutting feed state and the absolute value of the X-axis absolute coordinate value changes). After execution of S0, the analog voltage output is 0V. CNC

During reset and emergency stop, the analog voltage output remains unchanged.

Transmission parameter 107-110 parameter No.: the first spindle 1 to 4 gears (corresponding to M41 to M44) when the spindle maximum speed (output analog voltage is 10V) Transmission parameters of 111 parameters: the second spindle high spindle when the maximum speed (output analog voltage is 10V)

Constant linear speed control G96, constant speed control G97

Instruction format: G96 S ;(S0000 ~ S9999, leading zero can be omitted)

command function: constant line speed control is effective, given cutting line speed (min/min), cancel constant speed control. G96 is the G code of the mode. If the mode is G96, you may not enter G96.

Instruction format: G97 S ;(S0000 ~ S9999, leading zero can be omitted)

command function: cancel the constant linear speed control, constant speed control is effective, given the spindle speed (r/min). G97 is a modal G command. If the current mode is G97, G97 may not be entered. G97 is required.

Instruction format: G50 S ;(S0000 ~ S9999, leading zero can be omitted)

Command function: set the maximum spindle speed limit value (r/min) during constant linear speed control; If the coordinate value is commanded at the same time, set the current point as the program zero point.

G96 and G97 are modal instruction words of the same group, and only one can be valid. G97 is the initial instruction word, and when CNC is powered on, G97 is valid by default.

When turning a workpiece, the workpiece usually rotates with the spindle axis as the center line. The cutting point of the tool cutting the workpiece can be regarded as a circular motion around the spindle axis. The instantaneous rate of circumferential tangent direction is called cutting linear velocity (usually referred to as linear velocity). Different materials of the workpiece, different materials of the tool requirements of the line speed is different.

When the spindle speed analog voltage control function is effective, the constant linear speed control function is effective. In constant linear speed control, the spindle speed with the programmed trajectory (Ignoring tool length compensation) of the X-axis absolute coordinate value of the absolute value of the change, the X-axis absolute coordinate value of the absolute value increases, the spindle speed decreases, The absolute value of the X -axis absolute coordinate value is reduced, and the spindle rotation speed is increased so that the cutting linear speed is maintained at the S command value. The constant wire speed control function is used to cut the workpiece, which can keep the surface finish of the workpiece with varying diameter.

Linear speed = spindle speed x | X | x π ÷ 1000 (m/min)

spindle speed: rpm

| X |: the absolute value of the X-axis absolute coordinate value (diameter value), mm

π ≈ 3.14

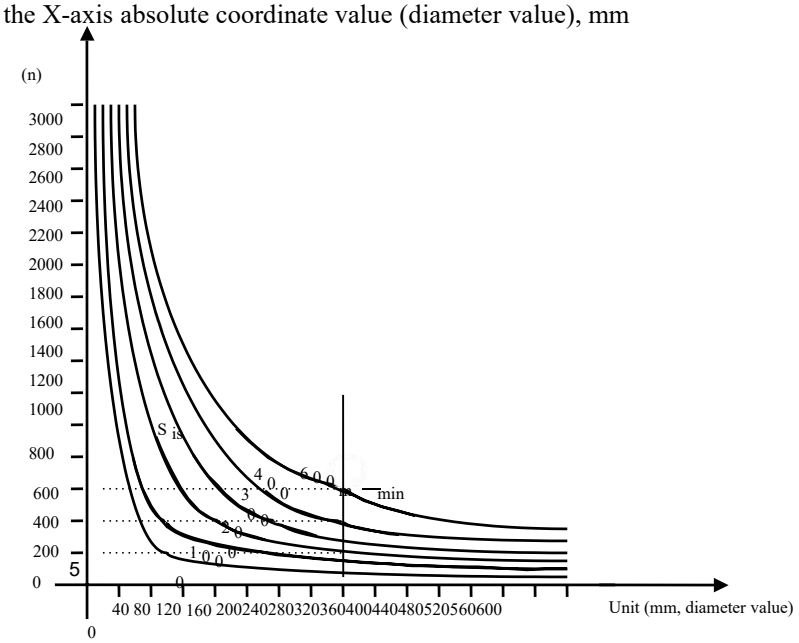


Figure 2-4

Constant line speed control, only in the cutting feed (interpolation) process with the programming track of the X axis absolute coordinate value of the absolute value of the change time change

Spindle speed; For the G00 fast movement, because there is no actual cutting, G00 execution process spindle speed remains unchanged, at this time the main

The shaft speed is calculated according to the linear speed of the end position of the program section, and directly becomes the end speed.

Constant line speed control, the workpiece coordinate system Z coordinate axis and spindle axis (workpiece rotation axis) must coincide, otherwise, the actual line speed will be with

The given line speed is inconsistent.

When the constant linear speed control is effective, G50 S_can limit the maximum rotation speed (rpm). When the linear speed and the X-axis coordinate value are calculated for the spindle

When the speed is higher than this limit spindle speed limit set by G50 S_, the actual spindle speed is the G50 maximum speed limit.

When the CNC is powered on, the maximum spindle speed limit value is not set, and the maximum spindle speed limit function is invalid. G50 S Defined Highest Turn

The speed limit value is maintained before re-assignment or power failure. The maximum speed limit function is valid in G96 state and G50 S in G97 state.

The set maximum spindle speed does not have a limiting effect, but the maximum spindle speed limit is still maintained.

Special attention is required: if G50 S0 is implemented, the spindle speed will be limited to 99 r/min during constant linear speed control (set by basic parameter 160 number);

Execute G50 S Set the maximum speed limit for constant line speed control.

The CNC drive parameter 116 number is the lower limit of the spindle speed when the constant line speed is controlled. When the spindle speed is lower than the line speed and the X axis coordinate value

This value is that the actual spindle speed is limited to the lower spindle speed limit.

example:

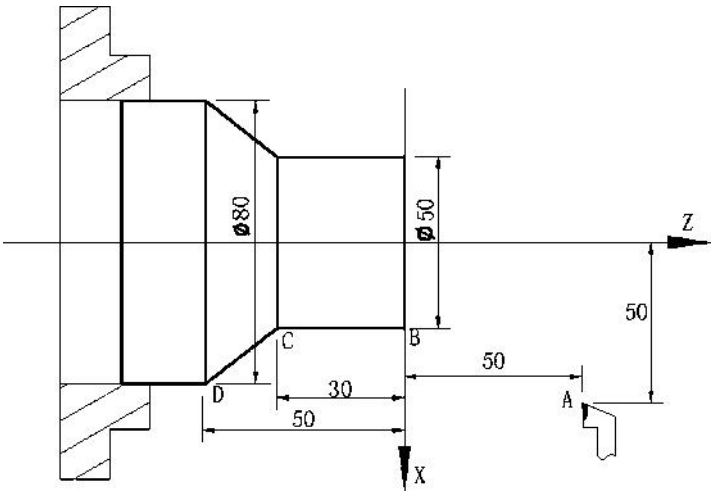


Figure 2-5

O0001	;	(Program Name)
N0010	M3 G96 S300;	(Spindle forward rotation, constant line speed control is effective, line speed is 300 m/min)
N0020	G0 X100 Z50;	(Quickly move to point A, the spindle speed is 955r/min during the movement)
N0030	G0 X50 Z0;	(Move quickly to point B, the spindle speed is 1910r/min during the movement)
N0040	G1 W-30 F200;	(From point B to point C, the spindle speed during cutting is 1910r/min)
N0050	X80 W-20 F150;	(From point C to point D, the spindle speed changes linearly from 1910r/min to 1194r/min)
N0060	G0 X100 Z100;	(Quickly return to point A, the spindle speed is 955r/min during movement)
N0110	M30;	(End of the program, turn off the spindle and coolant)
N0120	%	

Note **1**: In the **G96** state, the commanded **S** value is maintained even in the **G97** state. When returning to the **G96** state, its value is restored;

For example:

G96 S50; (**50 m/min** cutting speed)
G97 S1000 ;(**1000r/min** spindle speed)
G96 X3000 ;(**50 m/min** cutting speed)

Note **2**: machine locked (do **X**, **Z** when commanded for axis motion **X**, **Z** when the shaft does not move), the constant linear speed control function is still effective;

Note **3**: During thread cutting, the constant linear speed control function is invalid, but the relevant G0 positioning is effective. In order to ensure the accuracy of thread processing, the thread
Do not use constant linear speed control during cutting, but perform thread cutting in G97 state.

Note **4**: When changing from **G96** state to **G97** state, if there is no instruction **S** instruction (**r/min**) in the **G97** program segment, then the final rotation speed of **G96** state is used as the **G97** state **S** instruction, I .e. the spindle rotation speed remains unchanged;

Note **5**: In constant linear speed control, when the spindle speed calculated from the cutting linear speed is higher than the maximum speed of the current spindle gear (**CNC** parameters **No**.**107-110**), the spindle speed at this time is limited to the maximum speed of the current spindle gear.

Spindle magnification

When the spindle speed analog voltage control mode is effective, the actual speed of the spindle can be adjusted by the spindle magnification. The actual speed after the spindle magnification is adjusted is limited by the maximum speed of the current gear of the spindle, and is also limited by the minimum spindle speed limit value and the maximum spindle speed limit value under the constant linear speed control mode.

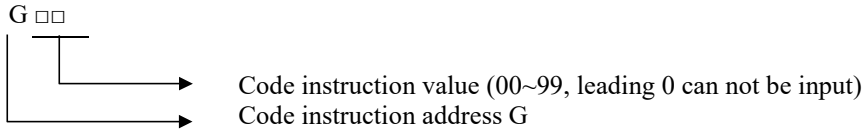
NC provides 8-level spindle magnification (50% ~ 120, 10% change per level)

G Code

Overview

The G code consists of the command address G and the following 1 to 2 bit code values, which are used to specify the movement mode of the tool relative to the workpiece and to set the coordinates.

See Table 3-1 for the list of G commands.



G-code instruction words are divided into groups 00, 01, 02, 03, 04. In addition to 01 and 00 group code can not be a common segment, the same program segment can enter several different groups of G-code instruction words, if two or more G-instruction words of the same group are entered in the same program segment, the last G-code command word is valid. Different groups of G instructions without common parameters (instruction words) can be in the same program segment, and the functions are valid at the same time and successively.

Order is irrelevant. If G-code commands other than those in Table 3-1 or G-code commands for optional functions are used, the system alarms.

Table 3-1G Code List

Code	Group	Function	Remarks
G00	01	Move fast	Initial state G command
G01		linear interpolation	Modal G Command
G02		Circular interpolation (counterclockwise)	
G03		Circular Interpolation (Clockwise)	
G32		thread cutting	
G90		axial cutting cycle	
G92		thread cutting cycle	
G94		radial cutting cycle	
G04	00	Suspension, quasi-stop	Non-modal G command
G28		Return to mechanical zero	
G50		Coordinate System Settings	
G51		Restore Workpiece Coordinate System Settings	
G65		macro instruction	
G70		Finishing cycle	
G71		Axial rough cycle	
G72		Radial rough cycle	
G73		Closed cutting cycle	
G74		Axial grooving multiple cycles	
G75		Radial grooving multiple cycles	
G76		Multiple thread cutting cycles	
G96	02	Constant line quick opening	Modal G Command
G97		Constant line speed off	Initial state G command
G98	03	Feed per minute	Initial state G command
G99		per redirection feed	Modal G Command
G40	04	Cancel nose radius compensation	Initial state G command
G41		Tip radius left compensation	Modal G Command
G42		Neat Radius Right Compensation	

Modal, non-modal and initial state

G-code instructions are divided into 00, 01, 02, 03, 04 groups. Among them, 00 groups of G instructions are non-modal G instructions, other groups of G code instructions are modal G code instructions, and G00, G97, G98 and G40 are initial G code instructions.

After a G -code instruction is executed, its defined function or state remains valid until it is changed by another G instruction in the same group. Such a G instruction is called a modular **G** -code instruction. After the modal G code instruction is executed, before its defined function or state is changed, when the subsequent program segment executes the G code instruction, it is not necessary to re-enter the G code instruction.

After the G code instruction is executed, the function or state defined by it is valid once. Each time the G code instruction is executed, the G generation code instruction word must be re-entered. This G code instruction is called a non-modal G code instruction.

After the system is powered on, a modal G-code instruction that is valid without executing its function or state is called an initial **G** -code instruction. When G code instruction is not input after power-on, it is executed according to the initial state G code instruction. GSK980TD1 initial state code instructions are G00, G40, G97, and G98.

Omission input of instruction word

In order to simplify programming, Table 3-2 listed code instruction words with the execution of the instruction value to maintain the characteristics, if the previous program segment has included these instruction words, in the subsequent program segment need to use the instruction value of the same, the same meaning of the instruction word, can not be entered.

Table 3-2

Instruction Address	functional significance	Initial value at power-on
U	G71 middle cutting depth	
U	X-axis retreating distance in G73	
W	G72 middle cutting depth	
W	G73 Middle Z-axis Regression Distance	
R	Regression of G71 and G7 cycle 2	
R	Number of Rough Cycle in G73	
R	Regression after cutting in G74, G75	
R	G76 Medium Finishing Allowance	
R	G90, G92, G94, G76 middle taper	
(G98)F	Sub-feed speed (G98)	
(G99)F	Transfer feed speed (G99)	
F	Metric thread pitch (G32, G92, G76)	
I	Inch thread pitch (G32, G92,)	
S	Spindle speed designation (G97)	
S	Spindle line speed designation (G96)	
S	Spindle speed switch output	
P	Number of thread cutting finishing in G76; G76 Medium thread cutting Thread withdrawal width G76 medium thread cutting tip angle;	
Q	Minimum cut-in in G76	

Note 1: Instruction addresses with multiple functions (such as F, which can be used for a given feed per minute, feed per turn, metric thread pitch, etc.) are only allowed to be omitted when the same function definition instruction word is executed again after the instruction word is executed. For example, if G98 F is executed and thread instruction is not executed, the pitch must be input with F instruction word when metric thread processing is carried out;

Note 2: When the address X(U) and Z(W) are used for the coordinates of the end point of the given program segment, the input is allowed to be omitted. When the X(U) or Z(W) coordinate instruction word is not input in the program segment, the system takes the

absolute coordinates of the current X axis or Z axis as the coordinate value of the end point of the program segment;

Note 3: When using an instruction address not listed in Table 3-2, the corresponding instruction word must be entered and cannot be omitted.

Example 1:

```
O0001;
G0 X100 Z100; (fast move to X100 Z100; modal instruction word G0 is valid)
X20 Z30; (fast move to X20 Z30; modal instruction word G0 can omit input)
G1 X50 Z50 F300; (linear interpolation to X50 Z50, feed speed 300 mm/min; Mode command word
                  G1 is valid)
X100; (Linear interpolation to X100 Z50, feed speed 300 mm/min; No Z axis coordinates are input,
      Take the current coordinate value Z50;F300 keeps and G01 as modal instruction words, which
      can be omitted from input)
G0 X0 Z0; (fast move to X0 Z0, modal instruction word G0 valid)
M30;
```

Example 2:

```
O0002;
G0 X50 Z5; (fast to X50 Z5)
G04 X4;      (delay 4 seconds)
G04 X5;
M30;
```

Example 3 (power-up first run):

```
O0003;
G98 F500 G01 X100 Z100; (G98 per minute feed, feed speed 500 mm/min)
G92 X50 W-20 F2 ;    (F value must be input for thread cutting)
G99 G01 U10 F0.01;    (F value must be input for every feed of G99)
G00 X80 Z50 M30;
```

Related Definitions

In the description of the following contents of this manual, the meanings of relevant words (characters) without special instructions are as follows:

Starting point: the position of the current program segment before running;

End point: the position after the execution of the current program segment;

X: X axis absolute coordinates;

U: End point and start point X axis absolute coordinate difference;

Z: Z axis absolute coordinate;

W: End point and start point Z axis absolute coordinate difference;

F: Cutting feed speed.

Quick positioning G00

Instruction format: G00 X(U) Z(W) ;

Command function: X axis and Z axis move from the starting point to the end point at the same time, as shown in Figure 3-1.

The two axes move at their own independent speeds, the short axis reaches the end first, and the long axis moves the remaining distance independently, and its synthetic trajectory is not necessarily a straight line.

Instruction description**:** The G00 is the initial G code instruction.

The values of X, U, Z and W are in the range of -9999.999 ~ +9999.999mm;

X(U) and Z(W) can omit one or all. When one is omitted, it means that the axis's coordinate values are consistent with those of the start point and the end point. At the same time, omitting indicates that the end point and the starting point are at the same position. When X and U, Z and W are in the same program segment, X and Z are valid, and U and W are invalid.

Command trace diagram:

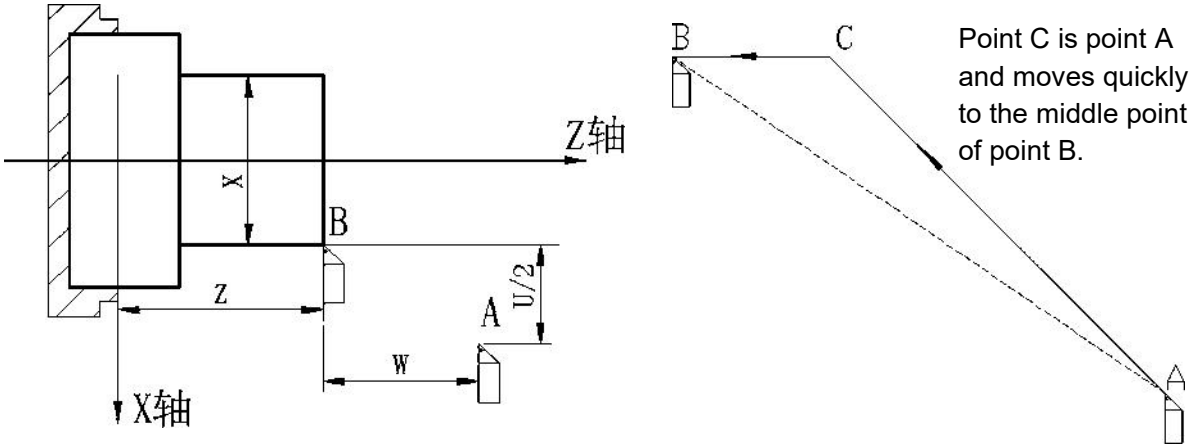


Figure 3-1

The X, Y, Z, A, B and C axes have their respective fast moving speeds respectively set by the system transmission parameters No. 72-77. The actual moving speed can be adjusted by the fast magnification key on the machine tool panel.

Example: The tool moves quickly from point A to point B. Figure 3-2

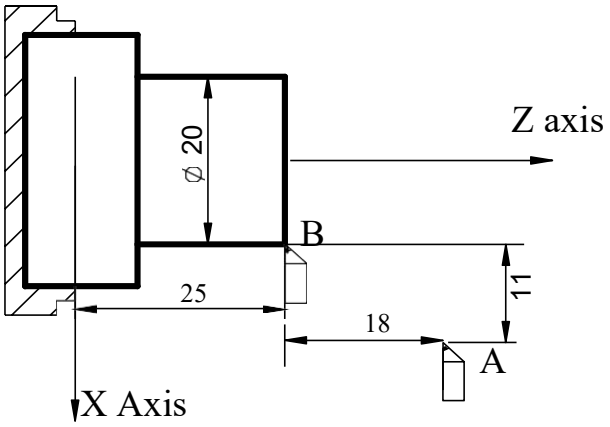


Figure 3-2

G0 X20 Z25;
G0 U-22 W-18 (absolute coordinate programming);
G0 X20 W-18 (relative coordinate programming);
G0 U-22 Z25 (mixed coordinate programming); (mixed coordinate programming)

Linear Interpolation G01

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

Command function: the motion trajectory is a straight line from the start point to the end point. The trajectory is shown in Figure 3-3. Instruction description**:** The G01 is the modal G code instruction.

The values of X, U, Z and W are in the range of -9999.999 ~ +9999.999mm;

One or all of X(U) and Z(W) can be omitted. When one is omitted, it means that the coordinate values of the start and end points of the axis are consistent;

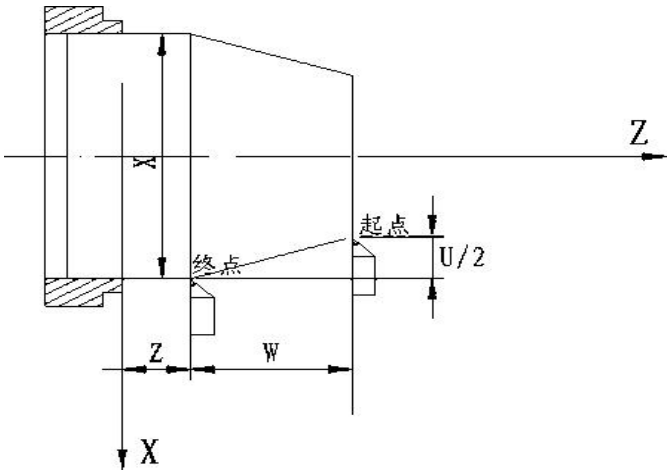
Also omitted indicates that the end point and the start point are at the same position.
The F command value is the vector synthesis speed of the instantaneous speed in the X -axis direction and the Z-axis direction, and the actual cutting feed speed is
The product of the given multiplier and the F command value;
After the F command value is executed, the command value is held until a new F command value is executed. Other G instructions described later make

Command Function	G98 (mm/min)	G99 (mm/rev)
Value range	1 to 8000	0.001~500

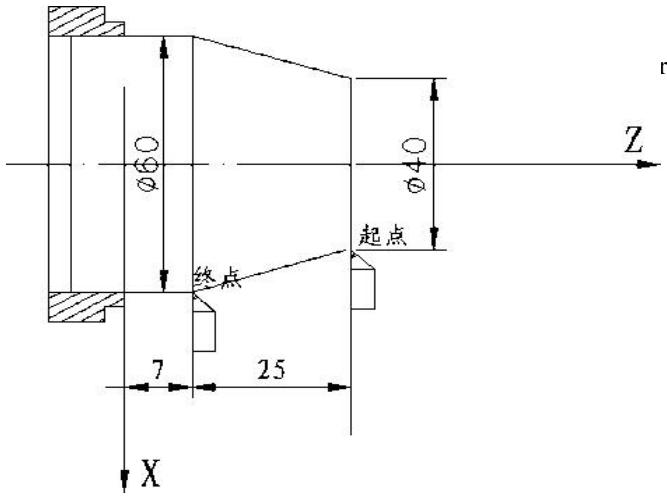
used
When the function of F instruction word is the same, it will not be described in detail.

The value range is as follows:

Command trace diagram:



Example: Program Instructions for Cutting from Diameter Φ40 to Φ60, Figure 3-4



rocedure:

G01 X60 Z7 F500;
G01 U20 W-25; (relative value programming)
G01 X60 W-25; (mixed programming)
G01 U20 Z7; Mixed programming

Circular Interpolation G02, G03

instruction format:

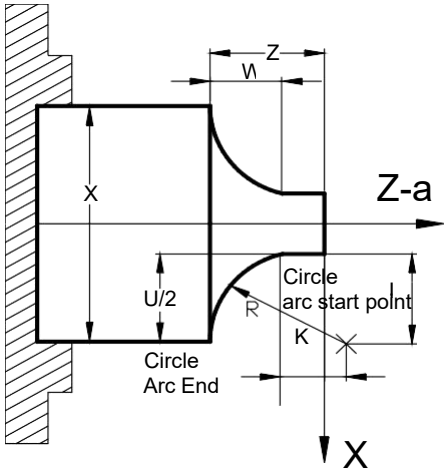
G02 / G03 } X（U）_ Z（W）__ { R__ / I K__

Command function:

G02 command trajectory from the start point to the end of the clockwise (after the tool holder coordinate system)/counterclockwise (before the tool holder coordinate system) arc. The trajectory is shown in Figure 3-5.

G03 command motion trajectory from the start point to the end point of the counterclockwise (rear tool seat coordinate system)/clockwise (front tool seat coordinate system) arc. The trajectory is shown in Figure 3-6.

Command trace diagram:



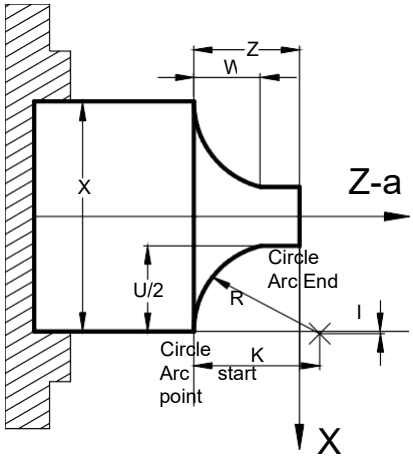


Fig. 3-5 G02 trajectory diagram fig. 3-6 G03 trajectory diagram instruction description: G02 and G03 are modal g instructions;

R is the arc radius, the value range -9999.999~9999.999mm;

I is the difference between the center of the circle and the starting point of the arc in the X direction, expressed by the radius, and the value range is -9999.999~9999.999mm;

K is the difference between the center of the circle and the starting point of the arc in the Z direction, and the value range is -9999.999~9999.999mm.

When the center of the arc is specified by addresses I and K, it corresponds to the X and Z axes respectively, and I and K represent the vector components from the starting point of the arc to the center of the circle, is the incremental value; as shown in Figure 3-6-1.

I = center coordinate X-X coordinate of arc starting point;

K = center coordinate Z-Z coordinate of arc starting point;

I and k are signed according to the direction, and the directions of I and k are the same as the directions of x and z axes, then positive values are taken; Otherwise, negative values are taken. the system will automatically adjust the position of the center of the circle

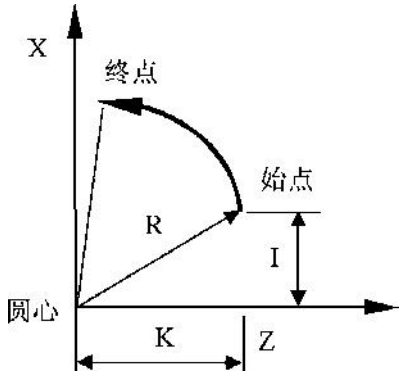


Figure 3- 6-1

Arc direction: the direction of G02/ G03 arc is defined. The front tool holder coordinate system and the rear tool holder coordinate system are opposite, as shown in Figure 3-7:

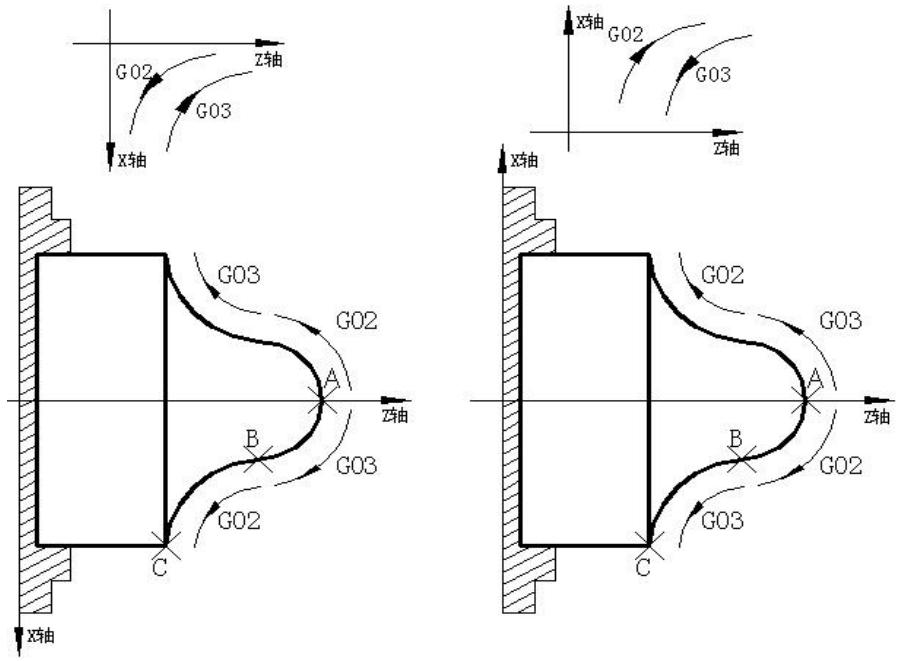


Figure 3-7

Note:

- When I = 0 or K = 0, it can be omitted; However, at least one instruction address I, K or R must be entered, otherwise the system will generate an alarm;
- When I, K and R are input at the same time, R is valid, and I and K are invalid;
- The R value must be equal to or greater than half of the starting point to the end point. If the end point is not on the arc defined by the R command, the system will generate an alarm;
- Address X(U), Z(W) can omit one or all; When one is omitted, it means that the starting point and ending point of the omitted axis are the same. At the same time, omission indicates that the end point and the starting point are at the same position. If I and K are used to instruct the center of the circle, the trajectory of executing G02/G03 instruction is full circle (360); When specified with R, it indicates a circle of 0 degree;
- R programming is recommended. When programming I and K are used, in order to ensure that the start and end points of arc motion are consistent with the specified values, the system presses the radius

  R = $\sqrt{I^2 + K^2}$ movement;

- If I and K values are used for programming, if the distance from the center of the circle to the end of the arc is not equal to R ((R = $\sqrt{I^2 + K^2}$

  Ensure that the start point and end point of the arc movement are consistent with the specified value. If the distance between the start point and the end point of the arc is greater than 2R, the system will give an alarm.

- When the r command is issued, it can be an arc greater than 180 and less than 180, an arc greater than 180 when the r is negative, and an arc less than or when the r is positive.

  an arc equal to 180 degrees;

Example: Program instruction for cutting an arc from diameter Φ45.25 to Φ63.06, Figure 3-8

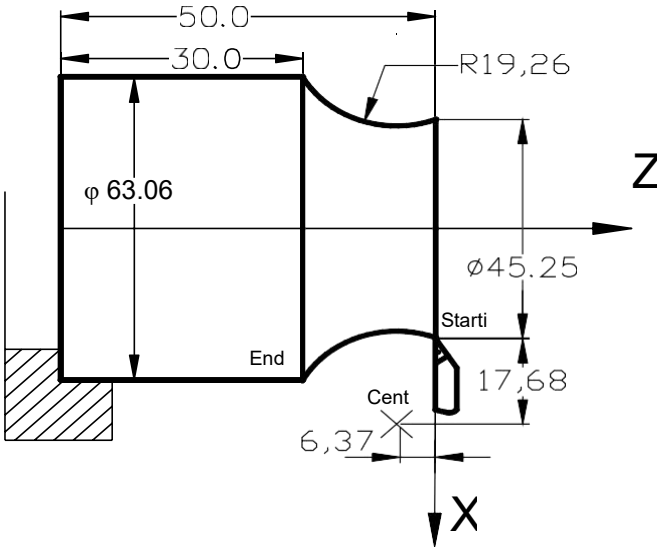


Figure 3-8

Procedure:

G02 X63.06 Z-20.0 R19.26 F300; or
G02 U17.81 W-20.0 R19.26 F300; or
G02 X63.06 Z-20.0 I17.68 K-6.37; or
G02 U17.81 W-20.0 I17.68 K-6.37 F300.

G02/G03 comprehensive instruction programming examples:

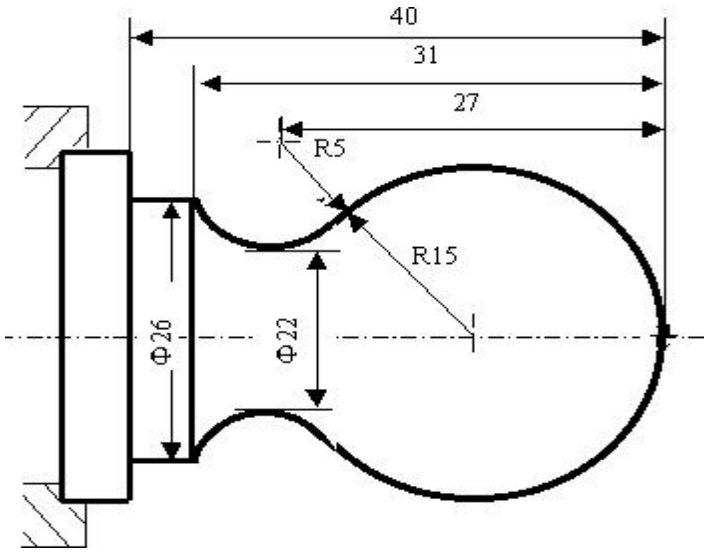


Figure 3-9 Arc Programming Example

- Procedure: O0001
  N001 G0 X40 Z5;
  N002 M03 S200;
  N003 G01 X0 Z0 F400; (near workpiece)
  N005 G03 U24 W-24 R15; (cutting R15 arc segment)
  N006 G02 X26 Z-31 R5; (cutting R5 arc segment)
  N007 G01 Z-40; (Cutting ϕ 26)
  N008 X40 Z5; (return to start)
  N009 M30;

PausecommandG04

Command format: G04 P;or

G04 X; or

G04 U; or

G04;

Command function: the movement of each axis stops, does not change the current G command mode and the maintained data and state, and delays for a given time, Then execute the next program segment.

Command description: G04 is a non-modal G command;

G04 delay time is specified by instruction word P, X or U;

P = 1~999999999

The range of X and U commands is 0.001 to 9999.999 seconds.

The time unit of the command value of the command word P, X or U is shown in the following table:

Table 3-3

Address	P	U	X
Unit	0.001 seconds	Seconds	Seconds

Note:

- When P, X and U are not entered or P, X and U specify negative values, it means that the program stops accurately between segments.
- P, X, U in the same program segment, P is valid; X, U in the same program segment, X is valid.
- During the execution of G04 instruction, the operation of feed holding is carried out, and the current delay time can be suspended only after the execution is completed.

Mechanical zero function

3.1.1 Machine tool first reference point G28

Instruction format: G28 X(U) Z(W) ;

Command function: start from the starting point, reach the middle point position specified by X(U) and Z(W) at a fast moving speed, and then return to the mechanical zero point.

Instruction description: G28 is a non-modal G code instruction;

X: the absolute coordinate of the X axis of the middle point;

U: the difference between the X-axis absolute coordinates of the intermediate point and the starting point;

Z: the absolute coordinate of the Z axis of the intermediate point;

W: The difference between the Z-axis absolute coordinates of the intermediate point and the starting point.

One or all of the instruction addresses X(U) and Z(W) can be omitted, as shown in the following table:

Table 3-4

Instruction	Function
G28 X(U)	X-axis returns to mechanical zero, Z-axis remains in place
G28 Z(W)	Z axis returns to mechanical zero, X axis remains in position
G28	Both axes remain in position and continue with the next 1 segment.
G28 X(U) Z(W)	X and Z axis return to mechanical zero at the same time

Command action process (as shown in Figure 3-10):

(1) Quickly locate from the current position to the middle point position of the command axis (point A → point B);

(2) Quickly locate from the middle point to the reference point (point B → point R);

(3) If the machine is not locked, the zero return light will be on when the reference point is returned.

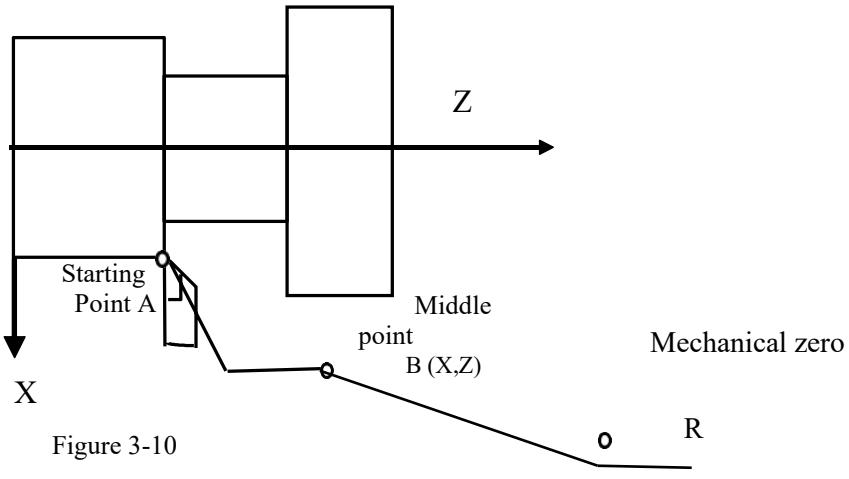


Figure 3-10

Reference point G30 of machine tool 2, 3 and 4

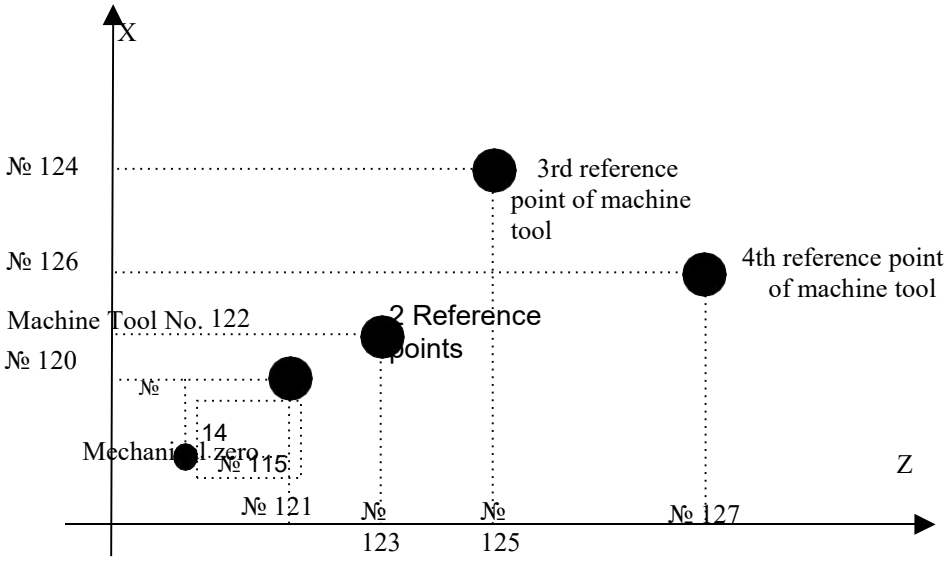
The mechanical zero point is a fixed point on the machine tool, which is determined by the zero point switch or the zero switch installed on the machine tool.

The machine tool reference point is the position after the value of the mechanical zero offset transmission parameter 130-135. When the set values of the transmission parameters 130-135 are all 0,

The machine reference point coincides with the mechanical zero point. The coordinates of the machine reference point are the values set by the transmission parameters 130-135. Perform mechanical zero return and G28 command zero return

The operation is to return to the reference point position of the machine tool.

The relationship between the mechanical zero point, the machine tool reference point, and the machine tool reference points 2, 3, and 4 in the machine tool coordinate system is shown in the following figure.



Command

G30 P2 X（U）_ Z（W） ;

G30 P3 X（U）_ Z（W） ;

G30 P4 X（U）_ Z（W） ;

Command function: start from the starting point, move at a fast speed to reach the middle point position specified by X(U) and Z(W) and then return

Reference points 2,3 and 4 of the machine tool. When returning to the second reference point of the machine tool, the command word P2 may be omitted.

Instruction Description:

G30 is a non-modal G command;

X: the absolute coordinate of the X axis of the middle point;

U: the difference between the X-axis absolute coordinates of the intermediate point and the starting point;

Z: the absolute coordinate of the Z axis of the intermediate point;

W: The difference between the Z-axis absolute coordinates of the intermediate point and the starting point.

One or all of the instruction addresses X(U) and Z(W) can be omitted, as shown in the following table:

Instruction	Function
G30 P n X(U)	X-axis returns to the nth reference point of the machine tool, and Z-axis remains in its original position
G30 P n Z(W)	Z axis returns to the nth reference point of the machine tool, and X axis remains in its original position
G30	Both axes remain in position and continue with the next 1 segment.
G30 P n X(U) Z(W)	X and Z axes return to the nth reference point of the machine tool at the same time

Note **1**: **n** in the table is **2**, **3** or **4**;

Note **2**: Return to the first **2** of the machine. **3**, the deceleration and zero point signals do not need to be detected during the **4** of the reference point.

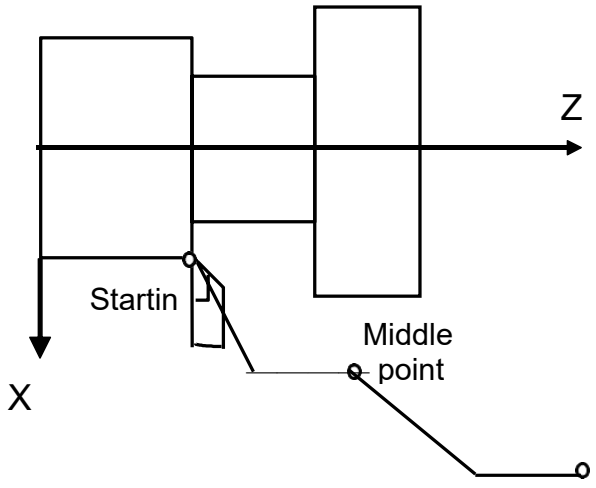
Command action process (as shown in the figure below, with reference point 2 of the machine tool):

(1) Quickly locate from the current position to the middle point position of the command axis (point A → point B);

(2) Locate from the middle point to the second reference point set by data parameters № 113 and № 123 at the speed set by data parameters № 122 (point B

→ R2 point);

(3) If the machine tool is not locked, the Bit0 bit and Bit1 bit of the reference point position return end signal ZP21 are high when returning to the reference point.



Reference point R2 of machine tool 2

Note **1**: the function of returning to the machine tool reference point in the first **2**, **3** and **4** reference point can only be used after returning to the machine tool reference point manually or after executing the **G28** command;

Note **2**: In the process from point **A** → point **B** and point **B** → point **R2**, the two axes move at their own independent speeds, so their trajectories are not necessarily
is a straight line;

Note **3**: Execute **G30** command to return to the first **2** and **3** of the machine tool. After **4** the reference point, the system cancels the tool length compensation;

Note **4**: If the machine tool is not equipped with a zero point switch, **G30** instruction machine tool **2**, **3**, **4** reference point operation shall not be executed; Note **5**: Return to the machine tool **2**, **3**, **4** reference point, do not set the workpiece coordinate system.

Workpiece coordinate system setting G50

Instruction format: G50 X(U) Z(W) ;

Command function: set the absolute coordinates of the current position, and establish the workpiece coordinate system (also called floating coordinate system) in the system by setting the absolute coordinates of the current position. After executing this instruction, the system will take the current position as the program zero point, and return to these 1 positions when executing the program zero point operation. After the workpiece coordinate system is established, the absolute coordinate programming inputs the coordinate values according to this coordinate system until the G50 is executed again to establish a new workpiece coordinate system.

Instruction description: G50 is a non-modal G instruction;

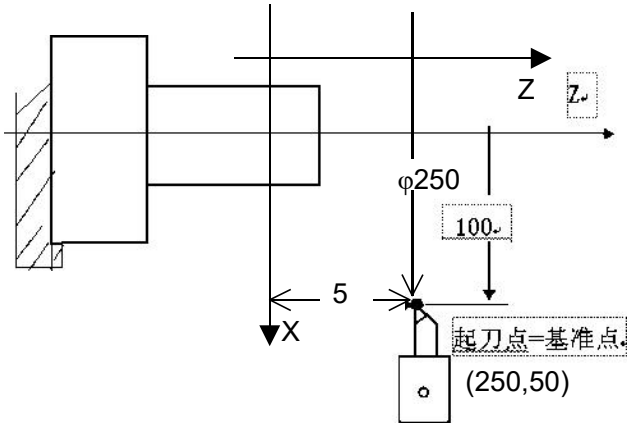
X: the new X-axis absolute coordinate of the current position;

U: the difference between the new X-axis absolute coordinates of the current position and the absolute coordinates before the execution of the command; Z: the new Z-axis absolute coordinates of the current position;

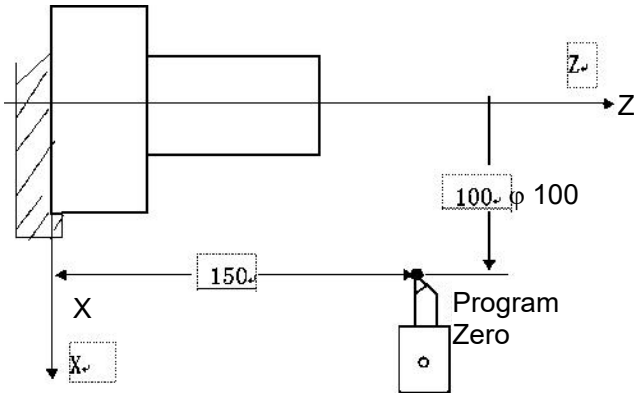
W: The difference between the new Z-axis absolute coordinates of the current position and the absolute coordinates before the execution of the command;

In the G50 command, when neither X(U) nor Z(W) is entered, the current coordinate value is not changed, and the current point coordinate value is set to the program zero point; if X(U) or Z(W) is not entered, the unentered coordinate axis remains the original program zero point.

Example:



Before using G50 to set the coordinate system



after using G50 to set the coordinate system as shown in Figure 3-11

As shown in Figure 3-11, when executing the instruction segment "G50 X100 Z150;", the workpiece coordinate system is established as shown in the figure, and the (X100 Z150) point is set as the program zero point.

Note: When the **T** function instruction is executed but the movement instruction is not executed, the coordinate system is set with **G50.** The absolute coordinate value displayed by the system is the coordinate value set by **G50** plus or minus the unexecuted tool compensation value, and this point is taken as the program zero point. For example.

Current state of tool compensation	Execute move command	Execute G50 X20 Z20 display coordinate values	No. 01 tool compensation value
T0100 or T0101	G0 X Z	X:20 Z:20	X:12 Z:23
	Move command not executed	Execute G50 X20 Z20 display coordinate values	
	※ ※ ※	X:8 Z:-3 or X:32 Z:43	

G51-To restore the workpiece coordinate system setting

code format]

G51

[description]]

G51 can only be a single segment and is not allowed to exist simultaneously with other instructions in the same program segment.

Restore the workpiece coordinate system and program reference point set in manual working mode. If the workpiece coordinate system and program reference point set in manual working mode are replaced by G50 instruction, G51 can be used to restore. After restoration, the system will use the old workpiece coordinate system and program reference point in both manual and automatic working modes until it is replaced.

Fixed Loop Instruction

In order to simplify the programming, the system provides only one program segment to complete the rapid movement positioning, straight line/thread cutting, and finally the rapid movement back.

G command for a single machining cycle at the starting point:
G90: Axial cutting cycle; G92: Thread cutting cycle; G94: Radial cutting cycle

The G92 Thread Cut Fix Cycle command is described in the Thread Function 1 section.

Axial cutting cycle G90

instruction format:

G90 X (U) _ Z(W) __ F ; (Cylindrical cutting)

G90 X (U) _ Z(W) __ F ; (Cone cutting)

Command function: starting from the cutting point, radial (X axis) feed, axial (Z axis or X, Z axis at the same time) cutting, to achieve cylindrical or conical surface

Cutting cycle.

Command description: G90 is a modal command;

Cutting starting point: the starting position of linear interpolation (cutting feed); Cutting end point: the end position of linear interpolation (cutting feed); X: X-axis absolute coordinate of cutting end point, unit: mm

U: the difference between the X-axis absolute coordinates of the cutting end point and the starting point, unit: mm; Z: the Z-axis absolute coordinates of the cutting end point, unit: mm; W: the difference between the Z-axis absolute coordinates of the cutting end point and the starting point, unit: mm;

R: The difference between the absolute coordinates of the X axis of the cutting start point and the cutting end point (radius value), with direction. When the signs of R and U are inconsistent, Requirements $|R| \le |U/2|$; When R = 0 or default input, cylindrical cutting is performed, as shown in Figure 3-12, otherwise conical cutting is performed,

As shown in Fig. 3-13; unit: mm.

Cyclic process: ① The X axis moves quickly from the starting point to the cutting starting point;

② linear interpolation from the cutting start point (cutting feed) to the cutting end point;

③ X-axis at the cutting feed speed, return to the X-axis absolute coordinates and the starting point of the same place;

④ The Z axis moves quickly back to the starting point and the cycle ends.

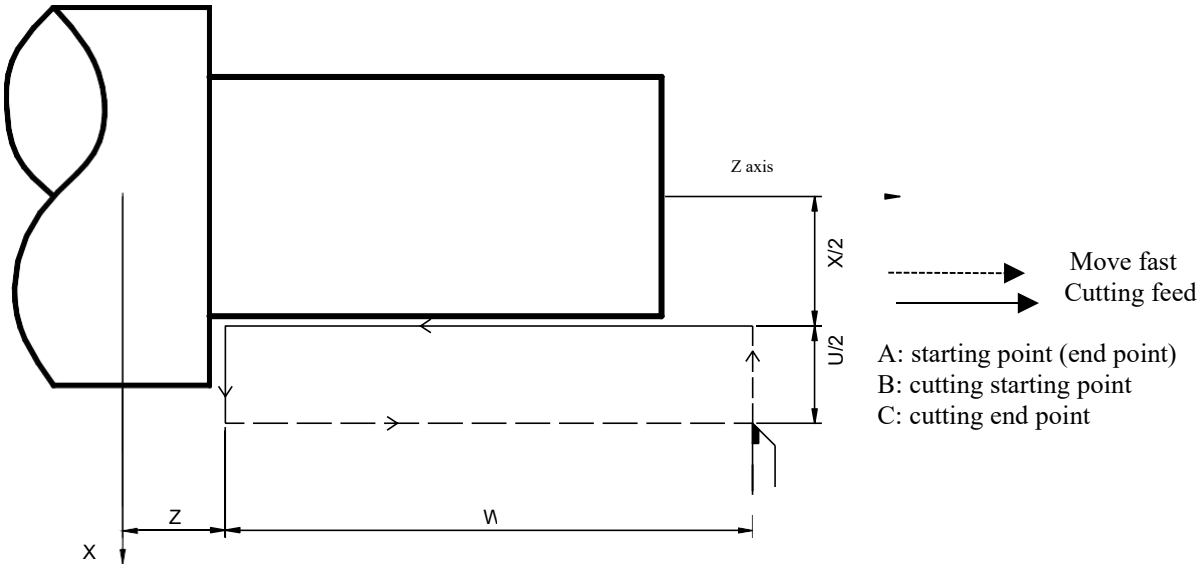


Figure 3-12

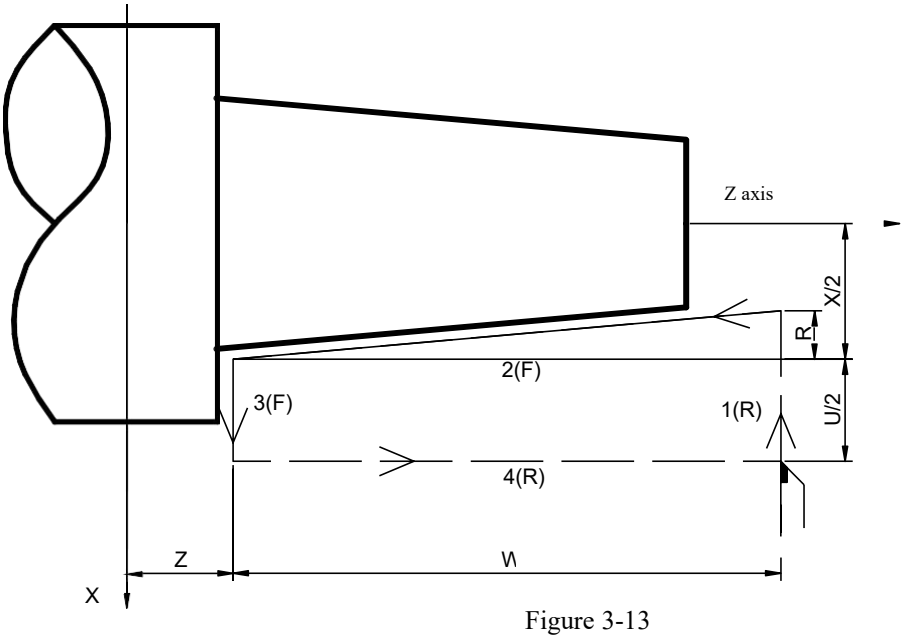
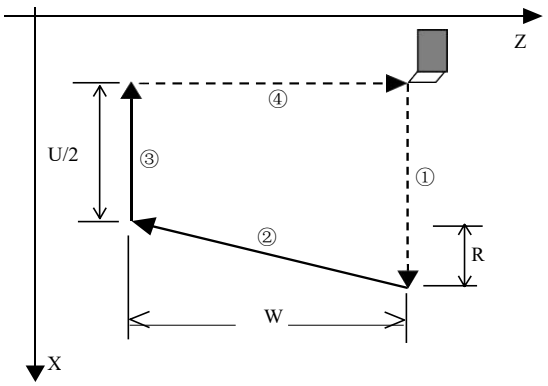


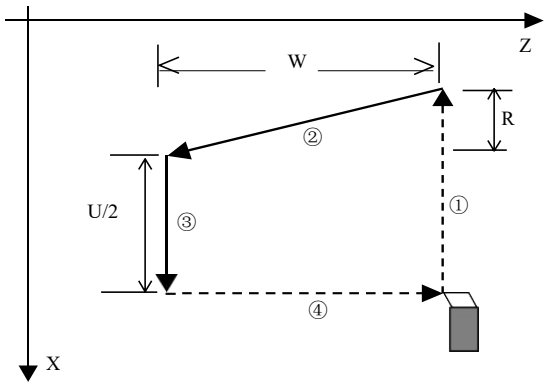
Figure 3-13

command trajectory: U, W and R reflect the relative positions of the cutting end point and the starting point, and the tool trajectory of U, W and R combined when the symbols are different, as shown in Figure 3-14.

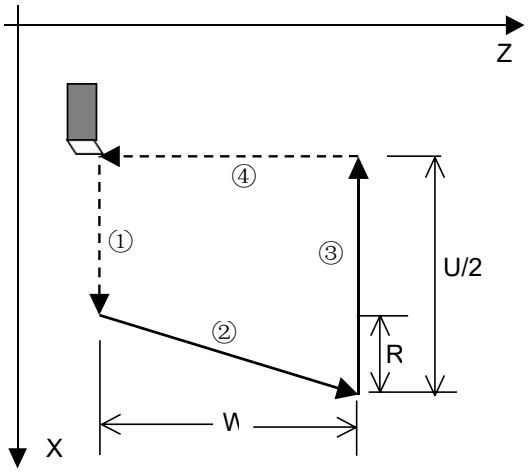
1) U>0,W<0,R>0



2) U<0,W<0,R<0



3） U>0，W>0，R<0 、｜R｜≤｜U/2｜



4） U<0，W>0，R>0、｜R｜≤｜U/2｜

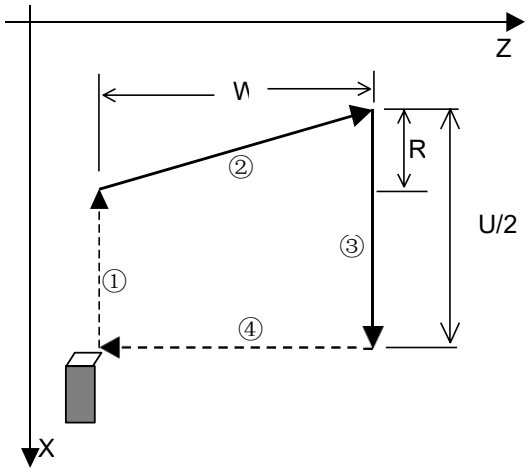


Figure 3-14

Example: Figure 3-15, Blank
Φ125×110

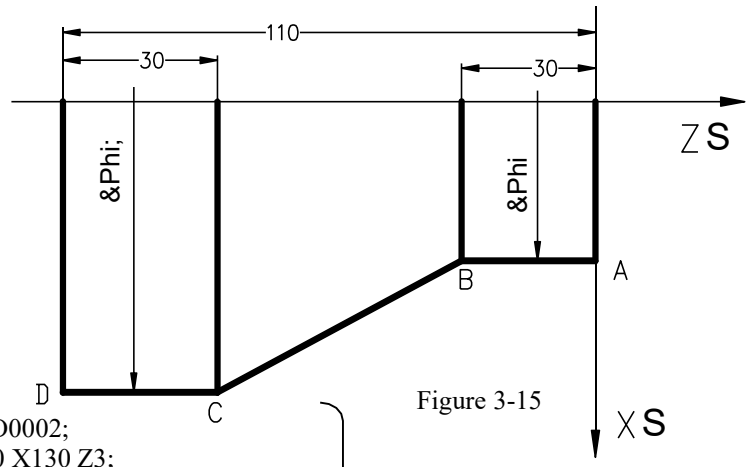


Figure 3-15

```
Procedure: O0002;
M3 S300 G0 X130 Z3;
G90 X120 Z-110 F200;                    (A→D，120 cutting)
X110 Z-30; X100;
X90; X80; X70; X60;                     (A→B , Φ60 cutting, six feed cycles
G0 X120 Z-30;                           Cutting, 10mm per feed)
G90 X120 Z-44 R-7.5 F150;
Z-56 R-15                               ,（B→C taper cutting, 4 feed cycles
Z-68 R-22.5 Z-80                        cutting)
R-30 M30;
```

Radial cutting cycle G94

Instruction format:

G94 X(U)	Z(W)	F ; (end cutting)	
G94 X(U)	Z(W) R	F ;(taper end cutting)	

Command function: starting from the cutting point, axial (Z axis) feed, radial (X axis or X, Z axis at the same time) cutting, to achieve end face or cone. The start and end points of the command are the same for the cutting cycle.

Instruction description: G94 is a modal instruction;

Cutting starting point: the starting position of linear interpolation (cutting feed), unit: mm; Cutting end point: the end position of linear interpolation (cutting feed), unit: mm; X: absolute coordinate of the X axis of the cutting end point, unit: mm; U: the difference between the absolute coordinate of the cutting end point and the X axis of the starting point, unit: mm; Z: absolute coordinate of the Z axis of the cutting end point, unit: mm; W: Z-axis absolute coordinate difference between the cutting end point and the starting point, unit: mm;

R: The difference between the absolute coordinates of the Z axis between the cutting start point and the cutting end point. When the symbols of R and W are different, $|R| \leq |W|$,

Radial linear cutting is shown in Figure 3-16, and radial taper cutting is shown in Figure 3-17, unit: mm.

Cyclic process: ① The Z axis moves quickly from the starting point to the cutting starting point;

② linear interpolation from the cutting start point (cutting feed) to the cutting end point;

③ Z-axis at the cutting feed speed (opposite to the direction of ①), return to the Z-axis absolute coordinates and the starting point of the same place;

④ The X-axis moves quickly back to the starting point and the cycle ends.

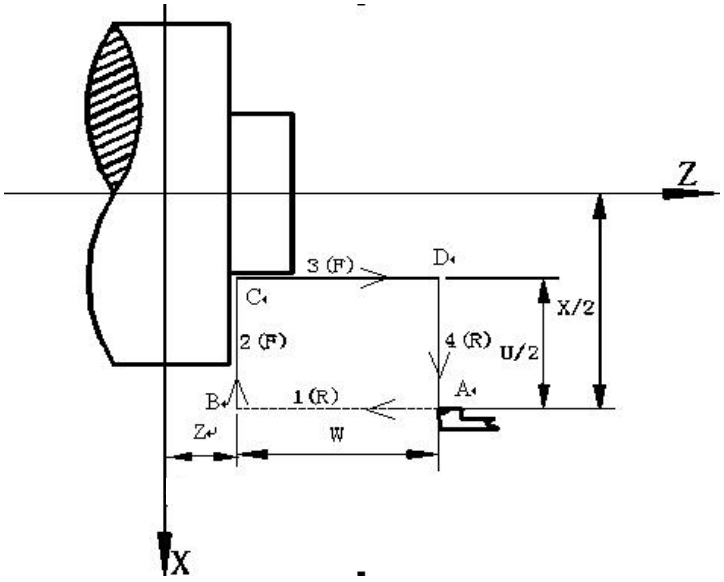


Figure 3-16

Fast cutting

cutting feed

A: starting point (end point) B: cutting starting point

C: cutting end point

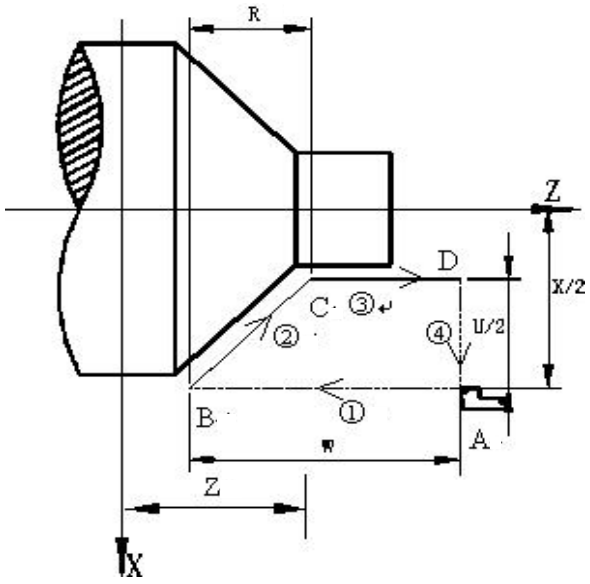
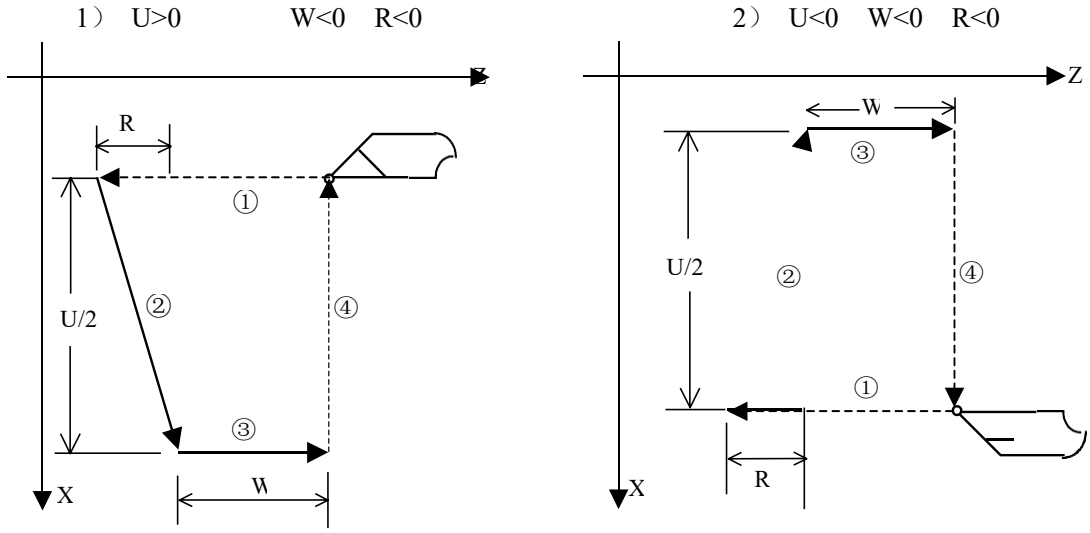


Figure 3-17

Command path: U, W, R reflect the relative position of the cutting end point and the starting point, and the tool path of U, W, R combined when the symbols are different,

as shown in figure 3-18:



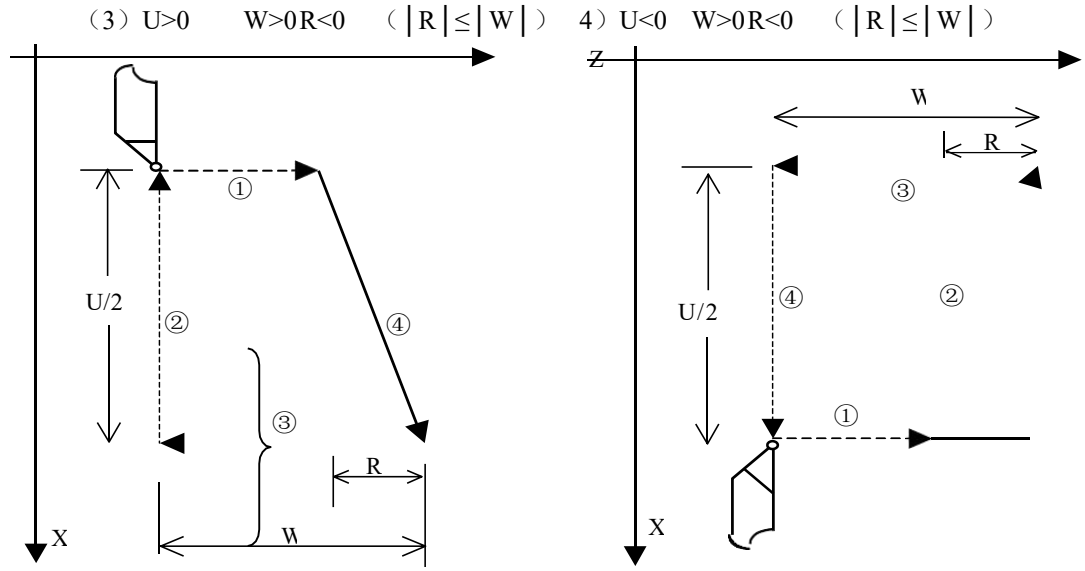


Figure 3-18

Example: Figure 3-19, Blank **Φ** 125 × 112

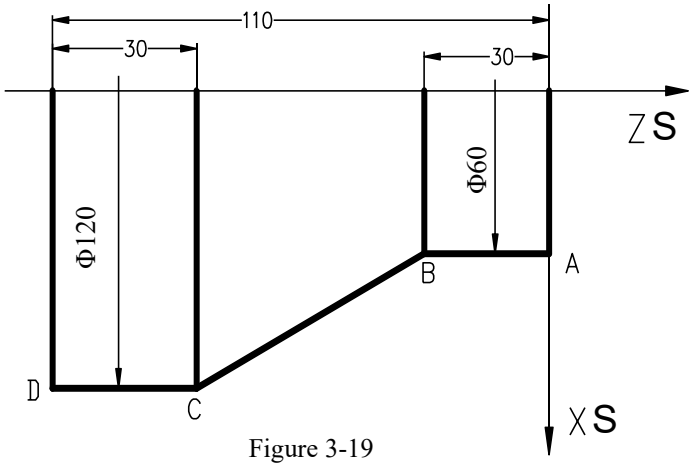


Figure 3-19

```
Procedure: O0003;
G00 X130 Z5 M3 S1;
G94 X0 Z0 F200                    } ending face cutting
X120 Z-110 F300                   } Outer circle &Phi;120 cutting)
G00 X120 Z0
G94 X108 Z-30 R-10
X96 R-20
X84 R-30                          (C→B→A, Φ60 cutting)
X72 R-40
X60 R-50;
M30;
```

Considerations for Fixed Loop Instructions

1) In the fixed loop instruction, once X(U), Z(W) and R are executed, X(U) is given again without executing the new fixed loop instruction.

Z(W),R, X(U),Z(W),R command values remain valid. If G04 other than non-modal (00 group)

G command or G00, G01, G02, G03, G32, the command values held by X(U), Z(W), R are cleared.

2) When the fixed cycle instruction is executed in the input mode, after the operation is completed, the instruction must be re-entered before the same fixed cycle as the previous one can be executed.

3) the m, s and t instructions are used immediately after the next program segment of the fixed loop G9 0~ G94 instructions, and the G90 ~ G94 instructions will not execute the loop once more; When the next 1 program segment has only EOB(;), the fixed loop will repeat the previous loop action.

Example:...

```
N010 G90 X20.0 Z10.0 F400;
N011 ; (Repeat G90 once here)
...
```

4) In the fixed loop G9, G94 instructions, pause at the current point when executing pause; Perform a single-segment operation, then move to the end of the entire trajectory after a single-segment stop (the entire G90, G94 is a single segment).

multiple cycle instruction

The multiple cycle instructions include: axial rough turning cycle G71, radial rough turning cycle G72, closed cutting cycle G73, finishing cycle G70, axial groove cutting multiple cycle G74, radial groove cutting multiple cycle G75 and multiple thread cutting cycle G76. When the system executes these instructions, it automatically calculates the cutting times and cutting trajectories according to the data such as programming trajectory, feed amount and retreating amount, and performs the processing cycle of multiple feed → cutting → retreating → refeeding, thus automatically completing the rough and finish machining of the workpiece blank. The starting point and end point of the instructions are the same.

The G76 Multiple Thread Cutting Cycle instruction is described in the Thread Function 1 section.

Axial rough cycle G71

Command Format: G71 U(Δd) R(e) F S T(1)
G71 P(ns) Q(nf) U(Δu) W(Δw)(2)
n(ns).....;
.....;
....F;
....S;
....
.

N(nf .

Instruction meaning: The G71 instruction is divided into 3 parts:

⑴: a given rough car when the amount of cutting, the amount of withdrawal and cutting speed, spindle speed, tool function of the program segment;

⑵: Given the program segment interval that defines the fine car trajectory and the program segment of the fine car margin;

(3): Define the fine car trajectory of several consecutive program segments. When executing G71, these program segments are only used to calculate the trajectory of the rough car and are not actually executed.

The system automatically calculates the rough machining route according to the data such as the fine turning trajectory, the fine turning allowance, the amount of feed, the amount of retreating, etc., along the parallel to the Z axis.

Direction cutting, through multiple feed → cutting → retreating cutting cycle to complete the roughing of the workpiece. The starting and ending points of the G71 are the same. This directive applies to non-forming blank (bar stock) forming rough car.

Related definitions:

Finish turning trajectory: the workpiece finishing trajectory given by part (3) of the instruction (ns ~ nf program segment), the starting point of the finishing trajectory (I .e. the starting point of ns program segment) is the same as the starting point and ending point of G71, referred to as point a; The first segment (ns segment) of the finishing track can only be the rapid movement or cutting feed of the X axis, and the end point of ns segment is simply referred to as point B; the end point of the finishing trajectory (the end point of the nf program segment) is referred to as point C. The track of the finish car is point A → point B → point C. (see Figure 3-20)

Rough vehicle contour: the trajectory after the fine vehicle trajectory is offset according to the fine vehicle margin (Δu, Δw), which is the trajectory contour formed by executing G71. Points A, B and C of the finishing track correspond to points A', B 'and C' of the rough turning contour after offset, and the final continuous cutting track of G71 command is B' point → C' point.

Δd:X-axis cutting amount during rough driving, with a value range of 0.001~99.999 (unit: mm, radius value), unsigned, and the feed direction is ns he direction of movement of the program segment. U(Δd) After execution, the command value Δd keep, and put the basic parameters 136 no. Modify the value Δd× 1000 (unit: 0.001) mm. When U(Δd) is not entered, the value No. of the basic parameter 136 is used as the feed amount.

e X-axis retreating amount during rough driving, with a value range of 0~99.999 (unit: mm, radius value), unsigned, retreating direction is opposite to feeding direction. After R(e) is executed, the instruction value E is maintained, and the value of basic parameter 137-No is modified to e× 1000 (unit: 0.001mm). When R(e) is not input, the value of the number of the basic parameter 137 is used as the retreating amount.
ns: program segment number of the first program segment of the fine vehicle trajectory; nf: program segment number of the last program segment of the fine vehicle trajectory.

Δu:X-axis finishing allowance, value range -9999.999~9999.999 (unit: mm, diameter), signed, rough car contour relative to the finishing car track of the X-axis coordinate offset, namely: A' point and A point X-axis absolute coordinate difference. When U is not entered (Δu), the system press

Δu = 0 for processing, I .e., the roughing cycle X axis leaves no finishing allowance.

Δw:Z-axis finishing allowance, value range -9999.999~9999.999 (unit: mm), signed, rough car contour relative to the finishing car trajectory of the Z-axis coordinate offset, namely: A' point and A point Z-axis absolute coordinate difference. When W is not entered (Δw), the system will process according to Δw = 0, that is, the rough turning cycle Z axis does not leave finishing allowance.
F: cutting feed speed; S: spindle speed; T: tool number, tool offset number.

M ST, F: can be in the first G71 instruction or the second G71 instruction, can also be in the ns ~ nf program.

In the G71 cycle, the functions m, s, t and f of the program segment number between ns and nf are all invalid, only with the G70 fine car cycle program

The paragraph is valid. Instruction execution process: Figure 3-20.

① Move quickly from point A to point A', X -axis Δu, Z -axis Δw;

② Move from A' point X axis Δd (feed),ns is the program segment G0 according to the fast moving speed feed, ns is the program segment G1 according to

G71 cutting feed speed F feed, feed direction and point A → B direction consistent;

③ Z axis cutting feed to the rough contour, the feed direction and point B → C Z axis coordinate change consistent;

④ X axis, Z axis according to the cutting feed speed of retreating e(45 ° straight line), retreating direction and each axis feed direction is opposite;

⑤ Z -axis at a rapid speed of movement to return to the A' point Z-axis absolute coordinates of the same position;

⑥ If after the X-axis re-feed (Δd + e), the end point of the movement is still in the middle of the line between A' and B '(it does not reach or exceed B'),

X-axis re-feed (d + e) and then execute ③; If after X-axis re-feed (d + e), the moving end point reaches B 'point or exceeds A' point →B' line, X -axis feed to B' point, then execute ⑦;

⑦ Cut and feed from point B 'to point C' along the rough turning contour;

⑧ moves quickly from point C' to point a, G71 loop execution ends, and the program jumps to the next nf program segment for execution.

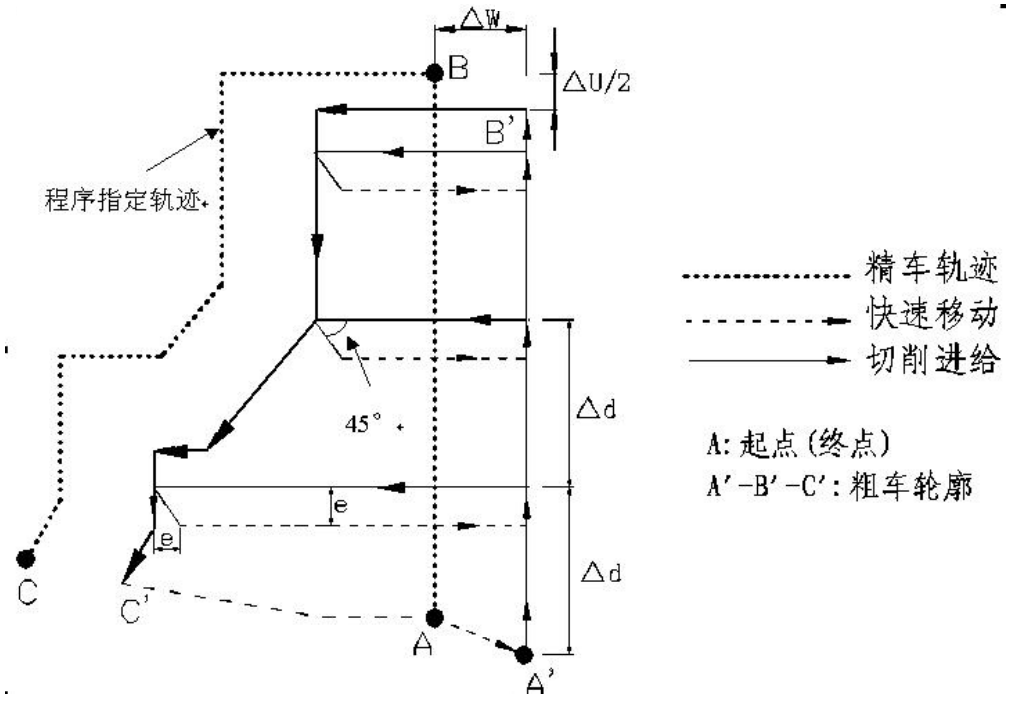


Figure 3-20 G71 Instruction Cycle Trajectory

Instruction Description:

- The ns ~ nf program segment must be written immediately after the G71 program segment. If it is written before G71 program segment, the system will automatically search for ns ~ nf program segment and execute it. after the execution is completed, the next 1 program of nf program segment will be executed in sequence, thus causing repeated execution of ns ~ nf program segment.
- When G71 is executed, the ns ~ nf program segment is only used to calculate the rough vehicle contour, and the program segment is not executed. The F, S and T instructions in the ns ~ nf program segment are invalid when executing the G71 cycle, and the F, S and T instructions in the G71 program segment are valid at this time. When executing the G70 finishing cycle, the f, s and t instructions in the ns ~ nf program segment are valid.
- Ns program segment can only be Z(W) instruction word G00, G01 instructions, otherwise alarm.
- Precision car trajectory (ns ~ nf program segment), the X axis, Z axis size must be monotonic change (always increase or always decrease).
- In ns ~ nf program segment, there can only be g functions: G00, G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,

  G42 instruction; no subroutine call instruction (e. g. M98/M99).
- The G96, G97, G98, G99, G40, G41, G42 commands are not valid for the G71 cycle and are valid for the G70 finishing cycle.
- During the execution of the G71 command, it is possible to stop the automatic operation and move manually, but to execute the G71 cycle again, you must return to the position before the manual movement. If you do not return to continue execution, the following run track will be misaligned.
- Execute the operation of feed hold and single program segment, and pause the program after running the end of the current track.
- △d,△u are all specified with the same address U, and the distinction is based on whether the program segment specifies P,Q instructions.
- The G71 instruction cannot be executed in the input mode, otherwise an alarm will be generated.
- When the compound loop instruction needs to be used multiple times in the same program, ns ~ nf does not allow the same program segment number.

Coordinate offset direction when leaving fine margin:

Δu and Δw reflect the coordinate offset and cut-in direction during finishing. There are 4 different combinations according to the symbols of Δu and Δw, as shown in Fig. 3-21,

In the figure, B→C is the track of fine car, B '→ C' is the outline of rough car, and A is the cutting point.

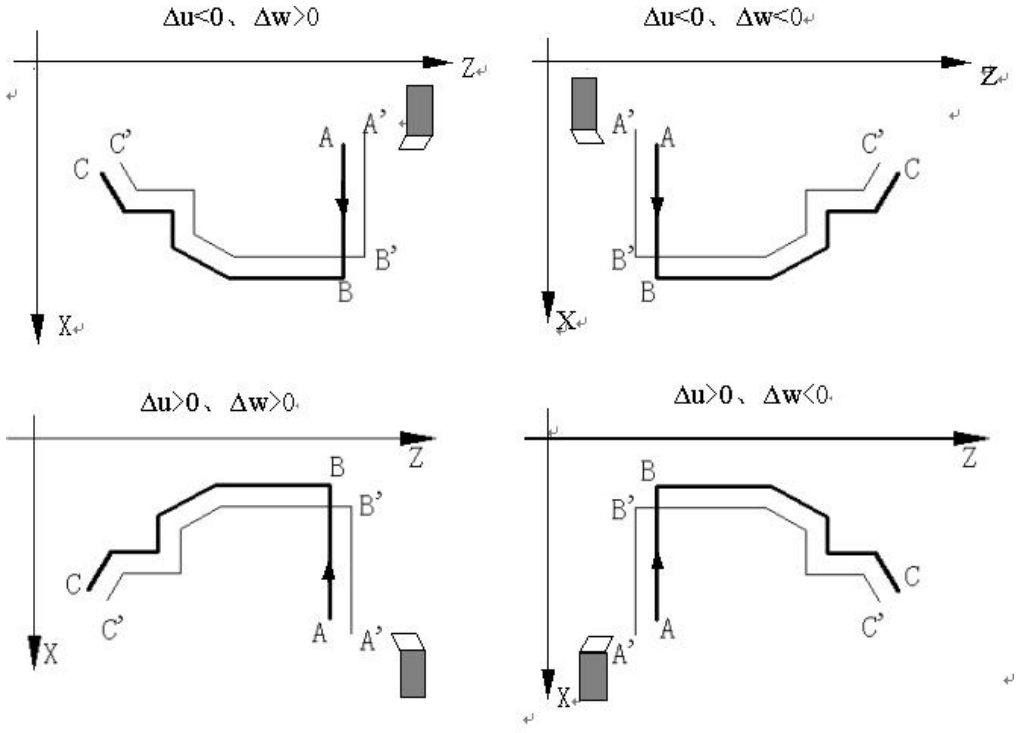


图 3-21

Example: Figure 3-22

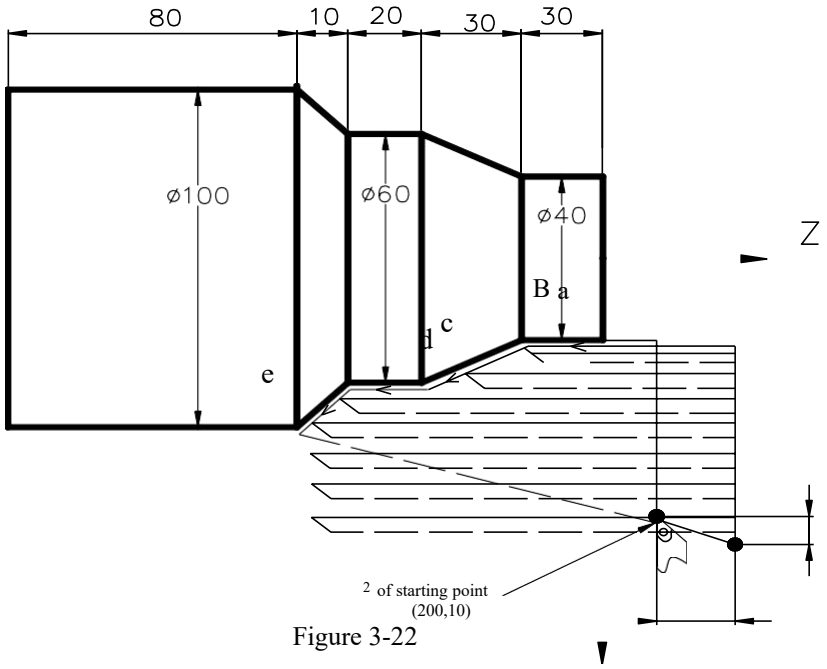


Figure 3-22

Procedure: O0004;

G00 X200 Z10 M3 S800; spindle (forward rotation, speed 800 r/min)

G71 U2 R1 F200; (Each cut depth 4mm, retreated 2mm [diameter])

G71 P80 Q120 U0.5 W0.2; (for a --- e rough turning, the allowance is 0.5mm in x direction and 0.2mm in z direction)

N80 G00 X40 S1200;
G00 Z0
G01 Z-30 F100 ; (a→b)
X60 W-30; (b→ c)
W-20; (c→d)
N120 X100 W-10; (d→e)

} Program section of finishing route a→b→c→d→e

G70 P80 Q120;（Precision turning of a -- e）

M30; (End of program)

Radial rough cycle G72

Command Format:

G72 W（Δd） R（e） F S T ； (1)

G72 P（ns） Q（nf） U（Δu） W（Δw）； (2)

N_（ns） . . . . . ;
. . . . . ;
. . . . F;
. . . . S;
. . . . ;
.
N__（nf）. . . . . ;

} (3)

Directive meaning: G72 directive is divided into 3 parts:

⑴: a given rough car when the amount of cutting, the amount of withdrawal and cutting speed, spindle speed, tool function of the program segment;

⑵: Given the program segment interval that defines the fine car trajectory and the program segment of the fine car margin;

(3): Define the fine car trajectory of several consecutive program segments, when executing G72, these program segments are only used to calculate the trajectory of the rough car, and are not actually executed.

The system automatically calculates the rough machining route according to the data such as the fine turning trajectory, the fine turning allowance, the amount of feed, the amount of retreating, etc., along the parallel to the Z axis.

Direction cutting, through multiple feed → cutting → retreating cutting cycle to complete the roughing of the workpiece, the starting point and end point of G72 are the same. This directive applies to non-forming blank (bar stock) forming rough car.

Related definitions:

Finish turning trajectory: the workpiece finishing trajectory given by part (3) of the instruction (ns ~ nf program segment), the starting point of the finishing trajectory (I .e. the starting point of ns program segment) is the same as the starting point and ending point of G72, referred to as point a; The first segment (ns program segment) of the finishing track can only be the rapid movement or cutting feed of the Z axis, and the end point of ns program segment is simply referred to as point B; the end point of the finishing trajectory (the end point of the nf program segment) is referred to as point C. The track of the finish car is point A → point B → point C.

Rough vehicle contour: the trajectory after the fine vehicle trajectory is offset according to the fine vehicle margin (Δu, Δw), which is the trajectory contour formed by executing G72. Points A, B and C of the finishing track correspond to points A', B 'and C' of the rough turning contour after offset. G72 commands the final continuous cutting track to be B' point → C' point.

Δd: The cutting amount of Z axis during rough turning, with a value range of 0.001~99.999 (unit: mm), without sign, and the feed direction is determined by the moving direction of ns program segment. W(Δd) After execution, the command value Δd keep, and put the basic parameters 136 no. Modify the value Δd×1000 (Unit: 0.001) mm. When W(Δd) is not entered, the value No. of the basic parameter 136 is used as the cutting amount. E: when the rough Z axis of the retreating amount, the value range is 0~99.999 (unit: mm), no sign, retreating direction and feed direction is opposite,

When R(e) is executed, the command value e is held, and the data basic 137 value number is changed to e× 1000 (unit: 0.001 mm). When R(e) is not input, the value of the number of the basic parameter 137 is used as the retreating amount.

ns: program segment number of the first program segment of the finisher track.

nf: program segment number of the last program segment of the fine vehicle trajectory.

Δu: the finishing allowance left by the X axis during rough turning, with a value range of -9999.999~9999.999 (X of rough turning contour relative to finish turning trajectory

Axis coordinate offset, I .e. the difference between the absolute coordinates of the X axis of point A' and point A, unit: mm, diameter, signed).

Δw: the finishing allowance left by the Z axis during rough turning, with a value range of -9999.999~9999.999 (Z of rough turning contour relative to finish turning trajectory

Axis coordinate offset, I .e. the difference between point A' and point A Z axis absolute coordinates, unit: mm, with symbol).

F: cutting feed speed; S: spindle speed; T: tool number, tool offset number.

M, S, T, F can be specified in the first G72 instruction or the second G72 instruction, or in the ns ~ nf program. In the G72 cycle, the functions of m, s, t and f of the program segment number between ns and nf are invalid, and only in the program segment with G70 fine turning cycle.

Instruction execution process: Figure 3-23.

◯ 1 Move quickly from the starting point A to the A' point, X -axis Δu, Z -axis Δw;

◯ 2 move from a' point z axis Δd (feed),ns is the program segment G0 according to the fast moving speed feed, ns is the program segment

G1 according to the G72 cutting feed speed F feed, feed direction and the A point to B point direction;

◯ 3 The X-axis cutting and feeding to the rough turning contour, and the feeding direction is consistent with the X-axis coordinate change of point B→C;

◯ 4 X axis, Z axis according to the cutting feed speed e(45 ° straight line), the retreating direction and each axis feed direction is opposite;

◯ 5 X -axis at a rapid speed of movement back to point A' Z-axis absolute coordinates of the same position;

◯ 6 If after the Z-axis re-feed (Δd + e), the end point of the movement is still in the middle of the line between A' and B '(it does not reach or exceed B'),

Z axis re-feed (Δd + e) and then execute ③; if after Z axis re-feed (Δd + e), the moving end point reaches B 'point

Or beyond the A'-B 'line, Z feed to B' point, and then execute ⑦;

◯ The 7 is cut along the rough turning contour from point B 'to point C';

◯ The 8 moves quickly from point C' to point A, the G72 loop execution ends, and the program jumps to the nf program segment's next program segment execution.

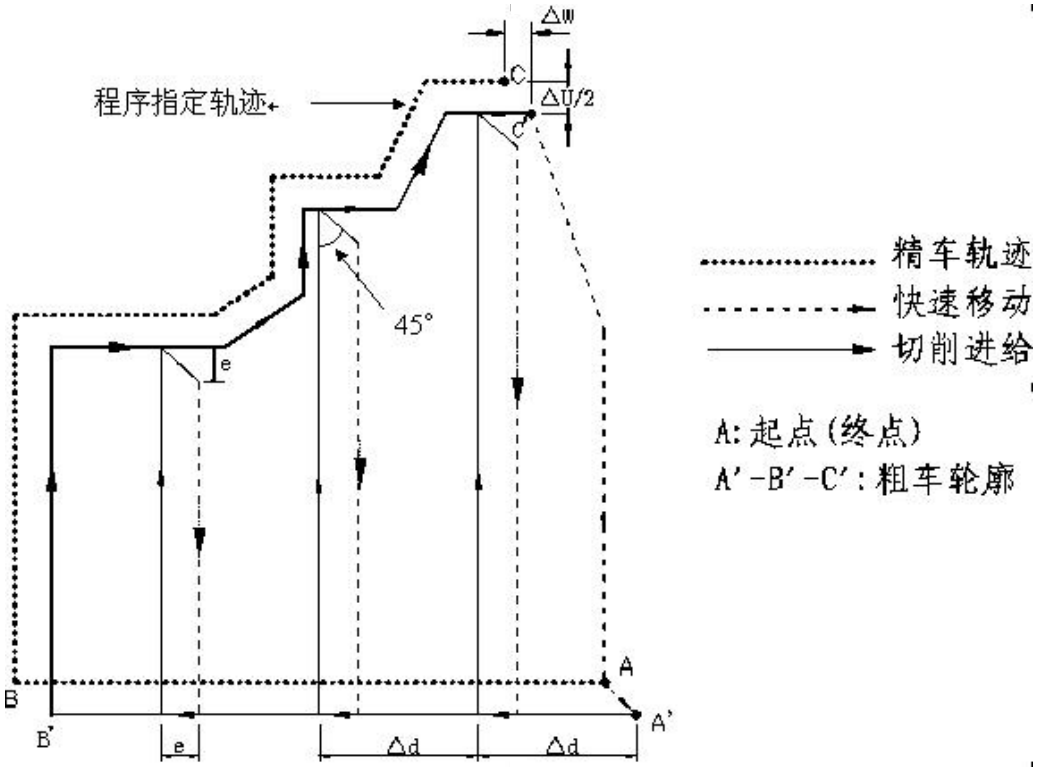


Figure 3-23

Instruction Description:

- The ns ~ nf program segment must be written immediately after the G72 program. If it is written before G72 program segment, the system will automatically search for ns ~ nf program segment and execute it. after the execution is completed, the next 1 of nf program segment will be executed in sequence, thus causing repeated execution of ns ~ nf program segment.
- When G72 is executed, the ns ~ nf program segment is only used to calculate the rough vehicle contour, and the program segment is not executed. The F, S and T instructions in the ns ~ nf program segment are invalid when executing the G72 cycle, and the F, S and T instructions in the G72 program segment are valid at this time. When performing the G70 finishing cycle, the F, S, and T instructions in the ns ~ nf program section are valid.
- Ns program segment can only be X(U) instruction words without G00 and G01 instructions, otherwise alarm.
- Trajectory (ns ~ nf program segment), the X-axis and Z-axis dimensions must be monotonic (always

increasing or decreasing);

- In ns ~ nf program segment, there can only be g functions: G00, G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,

  G42 instruction; no subroutine call instruction (e. g. M98/M99).
- The G96, G97, G98, G99, G40, G41, G42 commands are not valid for the G71 cycle and are valid for the G70 finishing cycle.
- During the execution of the G72 command, it is possible to stop the automatic operation and move manually, but to perform the G72 cycle again, you must return to the position before the manual movement. If you continue to execute without returning, the following running track will be misaligned;
- Execute the operation of feed hold and single program segment, and pause the program after running the end of the current track;
- △ d,△ w are all designated with the same address w, and the distinction is based on whether the program segment specifies P,Q instruction words;
- When the compound loop instruction needs to be used multiple times in the same program, ns ~ nf does not allow the same program segment number.

The G72 instruction cannot be executed in the input mode, otherwise an alarm will be generated.
Coordinate offset direction when leaving fine margin:

Δu and Δw reflect the coordinate offset and cut-in direction during finishing. There are 4 different combinations according to the symbols of Δu and Δw, as shown in Fig. 3-24,

In the figure, B→C is the track of fine car, B '→ C' is the outline of rough car, and A is the cutting point.

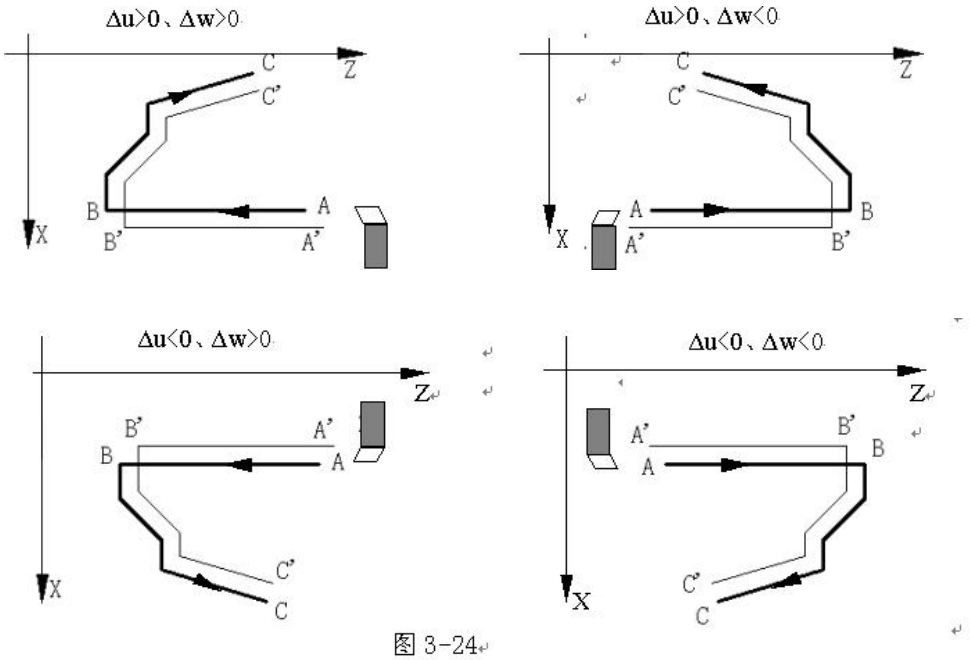


图 3-24

Example: Figure 3-25

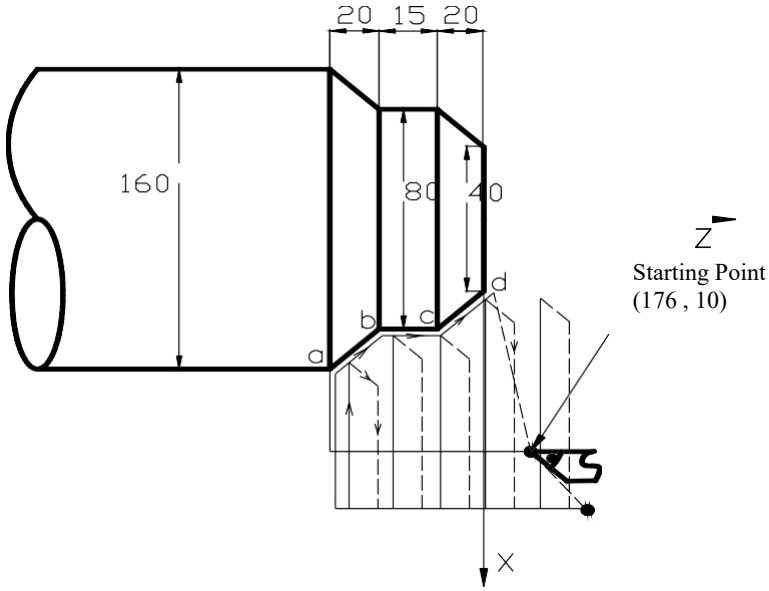


Figure 3-25

```
Procedure: O0005;
G00 X176 Z10 M03
S500                    (change to No.2 cutter, execute No.2 cutter deflection, main
                              shaft rotates forward, rotating speed 500)
G72 W2.0 R0.5 F300;     (Feed 2mm, retreated (0.5mm))
G72 P10 Q20 U0.2        (for a-d rough car, x leaves 0.2mm, z leaves 0.1mm allowance)
W0.1;

N10 G00 Z-55 S800 ;     (Quick Move)                          }
                                                              }
G00 X160;               (Positioning to point a)              }
                                                              }
G01 X80 W20;            (machining a-b)                       }  Finishing route section
                                                              }
W15;                    (processing b-c)                      }
                        (processing c-d)                      }
N20 X40 W20 ;                                                 }
                                                              }
G70 P010 Q020 M30;      (Finishing a-d)                       }
```

Closed cutting cycle G73

Instruction format: G73 U (Δ i) W (Δ k) R (d) F S T ; (1)

G73 P (ns) Q (nf) U (Δu) W (Δw); (2)

N_(ns) .....;
.....;
....F;
....S;
....; (3)

N (nf).....;

Instruction meaning: The G73 instruction is divided into 3 parts:

(1): a given amount of withdrawal, cutting times and cutting speed, spindle speed, tool function of the program segment;

(2): Given the program segment interval that defines the fine car trajectory and the program segment of the fine car margin;

(3): Define the fine car trajectory of several consecutive program segments. When executing G73, these program segments are only used to calculate the trajectory of the rough car and are not actually executed.

The system automatically calculates the rough turning offset, the single feed of the rough turning and the rough turning trajectory according to the data such as the fine turning margin, the amount of retreating, the number of cuts, etc. The trajectory of each cut is the offset of the fine turning trajectory, the cutting trajectory gradually approaches the fine turning trajectory, and the last cutting trajectory is the fine turning trajectory offset by the fine turning margin. The starting and ending points of G73 are the same. This directive applies to rough cars for forming blanks. G73 instruction is a non-modal instruction, and the instruction trajectory is shown in Figure 3-26.

Related definitions:

Finish turning trajectory: the workpiece finishing trajectory given by part (3) of the instruction (ns ~ nf program segment), the starting point of the finishing trajectory (I .e. the starting point of ns program segment) is the same as the starting point and ending point of G73, referred to as point a; The end point of the first segment (ns segment) of the finishing track is referred to as point B; The end point of the finishing track (the end of nf segment) is referred to as point c. The track of the finish car is point A → point B → point C.

Rough Trajectory: the 1 group offset trajectory of the fine turning trajectory. The number of rough turning trajectories is the same as the number of cutting times. A, Points B and C correspond to A of the rough track respectively $_n$, B $_n$, C $_n$ point (n is the number of cuts, the first cut is denoted as A_1, B_1, C $_1$ points, last expressed as A $_d$, B $_d$, C $_d$ points). The coordinate offset of the first cut relative to the finishing trajectory is

(Δi× 2 + Δu,Δw + Δk)(by diameter programming), the coordinate offset of the last cut relative to the fine turning trajectory is (Δu,Δw), and the coordinate offset of each cut relative to the previous cutting trajectory is:

$$(-\frac{\Delta i \times 2}{d-1} - \frac{\Delta k}{\text{d-1}})$$

Δi: X-axis coarse turning retreating amount, value range -9999.999~9999.999 (unit: mm, radius value, signed),Δi is equal to A x-axis coordinate offset (radius value) of $_1$ -axis point relative to A-$_d$ point, total X-axis cutting amount (radius value) is equal to | Δi |, and the cutting direction of X-axis is opposite to the sign of Δi: δi> 0, cutting in the negative direction of the X-axis during rough turning. The Δi command value is maintained after execution, and the value of the system basic parameter 138 number is modified to Δi × 1000 (unit: 0.001mm mm). When U(Δi) is not input, the value of the basic parameter 138 No. is used as the X-axis rough cutting amount.

Δk: Z-axis roughing tool withdrawal amount, value range -9999.999~9999.999 (unit: mm, signed) Δk is equal to A1 point relative A $_d$ A point of the Z axis coordinate offset, rough when the Z axis

of the total cutting amount is equal to | Δk |,Z axis of the cutting direction and Δk sign opposite:

Δk>0, cutting in the negative direction of the Z axis during rough turning. The Δk command value is maintained after execution, and the value of the basic parameter 139 number is modified to Δk × 1000 (unit: 0.001mm). When W(Δk) is not input, the value of the basic parameter No. 139 is used as the Z-axis rough turning relief amount.

d: the number of cutting, the value range is 1~9999 (unit: times),R5 means 5 times of cutting to complete the closed cutting cycle. R (d) command value is maintained after execution, and the value of the basic parameter 140 number is modified to d (unit: times). When R (d) is not input, the value of the number of basic parameter 140 is used as the number of times of cutting.

ns: program segment number of the first program segment of the finisher track.

nf: program segment number of the last program segment of the fine vehicle trajectory.

Δu:X-axis finishing allowance, value range -9999.999~9999.999 (unit: mm, diameter, signed), the last rough turning trajectory relative to the finishing turning trajectory of the X-axis coordinate offset, that is: A difference between the 1 point relative to the point of the X-axis absolute coordinate. Δu>0, the last rough track is offset in the positive direction of the X-axis relative to the fine track. When U(Δu) is not input, the system will handle according to Δu = 0, that is, the X axis of rough turning cycle will not leave finishing allowance.

Δw: finishing allowance of Z-axis, value range: -9999.999~9999.999 (unit: mm, with sign), Z-axis coordinate offset of the last rough turning track relative to the finishing turning track, I .e. difference of Z-axis absolute coordinate of 1 point A relative to point A. Δw>0, and the last rough track is offset in the positive direction of the Z axis relative to the fine track. When W(Δw) is not input, the system shall handle according to Δw = 0, I .e. no finishing allowance is left for Z axis of rough turning cycle.

F: cutting feed speed; S: spindle speed; T: tool number, tool offset number.

M, S, T, F instruction words can be specified in the first G73 instruction or the second G73 instruction, or in the program ns ~ nf. In the G73 cycle, the functions of M, S, T and F of the program segment number between ns and nf are invalid, and only in the program segment with G70 fine car cycle.

Instruction execution process: as shown in Figure 3-26.

① A→ A_1: Move quickly;

② For the first rough driving, $A_1 \rightarrow B_1 \rightarrow C_1$:

$A_1 \rightarrow B_1$:ns program segment is G0 according to the fast moving speed, ns program segment is G1 according to G73 specified cutting feed speed;

$B_1 \rightarrow C_1$: cutting feed.

③ $C_1 \rightarrow A_2$: fast movement;

④ For the second rough driving, $A_2 \rightarrow B_2 \rightarrow C_2$:

$A_2 \rightarrow B_2$:ns program segment is G0 according to the fast moving speed, ns program segment is G1 according to G73 specified cutting feed speed;

$B_2 \rightarrow C_2$: cutting feed.

⑤ $C_2 \rightarrow A_3$: fast movement;

.....

For the nth rough car, $A_n \rightarrow B_n \rightarrow C_n$:

$A_n \rightarrow B_n$:ns program segment is G0 according to the fast moving speed, ns program segment is G1 according to G73 specified cutting feed speed;

.....

$B_n \rightarrow C_n$: Cutting feed.

$C_n \rightarrow A_{n+1}$: fast movement;

.....

The last rough car, $A_d \rightarrow B_d \rightarrow C_d$:

$A_d \rightarrow B_d$:ns program segment is G0 according to the fast moving speed, ns program segment is G1 according to G73 specified cutting feed speed;

$B_d \rightarrow C_d$: Cutting feed. $C_d \rightarrow$ A:

Quickly move to the starting:

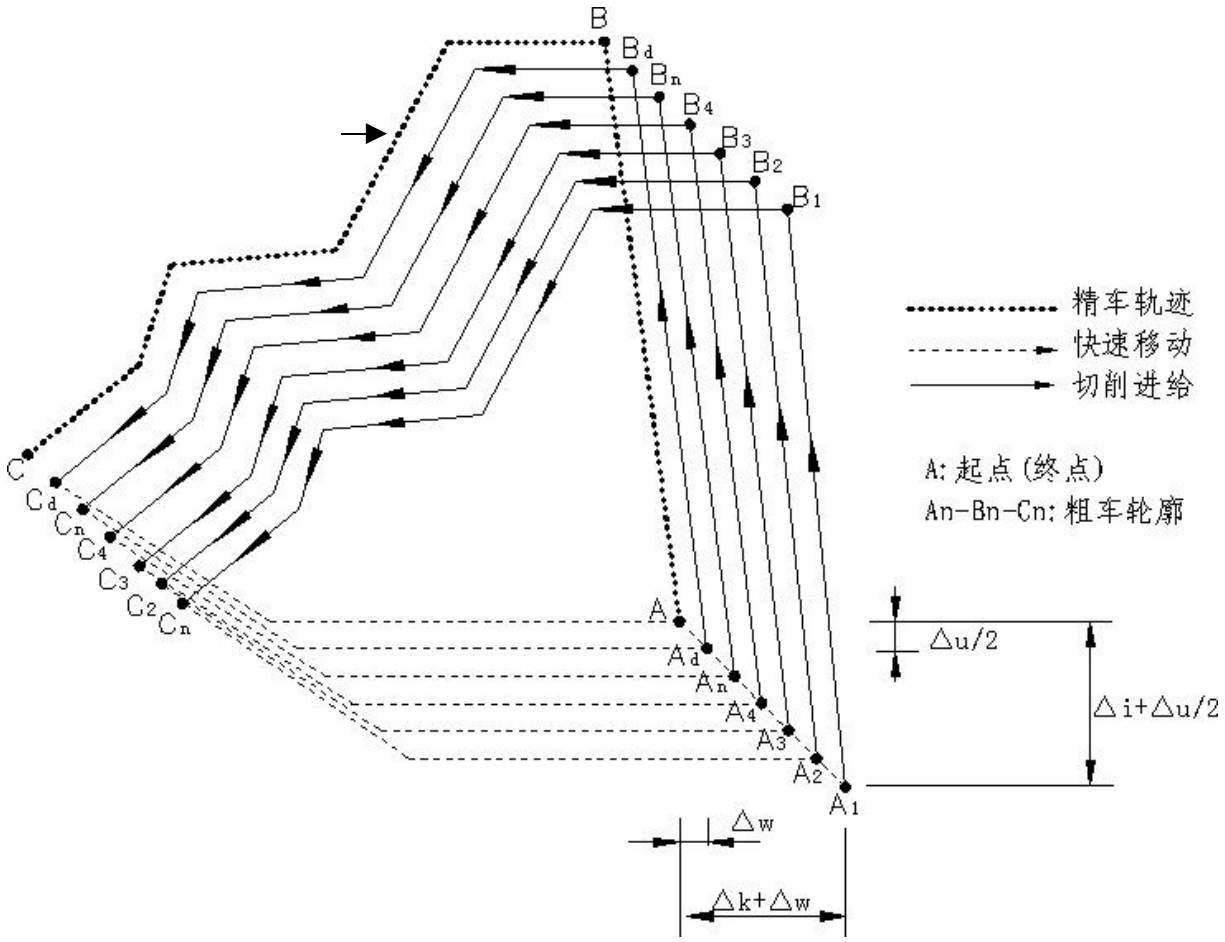


Figure 3-26 G73 command running track

Instruction Description:

- The ns ~ nf program segment must be written immediately after the G73 program segment. If the ns ~ nf program segment is written before the G73 program segment, the system can automatically search for the ns ~ nf program segment and execute it. after the execution is completed, the next 1 program of the nf program segment will be executed in sequence, thus causing repeated execution of ns ~
  nf program segment.
- When G73 is executed, the ns ~ nf program segment is only used to calculate the rough vehicle contour, and the program segment is not executed. The F, S and T instructions in the ns ~ nf program segment are invalid when G73 is executed, and the F, S and T instructions in the G73 program segment are valid at this time. When performing the G70 finishing cycle, the F, S, and T instructions in the ns ~ nf program section are valid.
- The ns program segment can only be G00 and G01 instructions. Ns program segment G02, G03 will generate an alarm.
- In the ns ~ nf program segment, there can only be the following g functions: G00, G01, G02, G03, G04, G96, G97, G98, G99, G40,

  G41, G42 instructions; Can not have the following M functions: subroutine call instructions (such as M98/M99).
- The G96, G97, G98, G99, G40, G41, and G42 commands are invalid for the G73 cycle and valid for the G70 finishing cycle.
- During the execution of the G73 command, it is possible to stop the automatic operation and move manually, but to execute the G73 cycle again, you must return to the position before the manual movement. If you do not return to continue execution, the following run track will be misaligned.
- Execute the operation of feed hold and single program segment, and pause the program after running the end of the current track.

- Δi,△u are all designated by the same address u, Δk,Δw are all designated by the same address w, and the distinction is based on whether the program segment specifies P,Q instruction words.
- The G73 instruction cannot be executed in the input mode, otherwise an alarm will be generated.
- When the compound loop instruction needs to be used multiple times in the same program, ns ~ nf does not allow the same program segment number.

Coordinate offset direction when leaving fine margin:

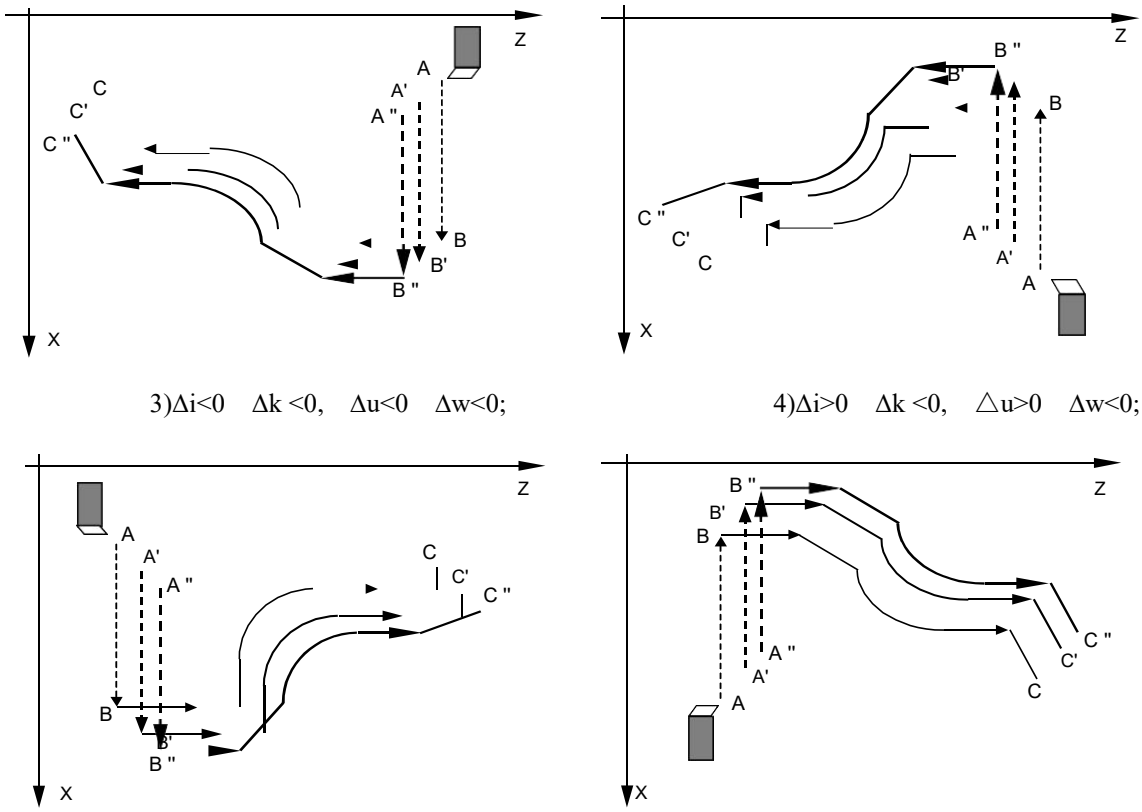
Δi and Δk reflect the coordinate offset and cut-in direction during rough steering, and Δu and Δw reflect the coordinate offset and cut-in direction during fine steering;

There can be many combinations of Δi, Δk, Δu, and Δw, and in general, Δi is usually consistent with the sign of Δu, and Δk is consistent with the sign of Δw,

There are 4 commonly used combinations, as shown in Figure 3-27. In the figure, A is the cutting point, B→C is the workpiece contour, and B '→ C' is the rough cutting contour.

B "→ C " is the fine vehicle trajectory.

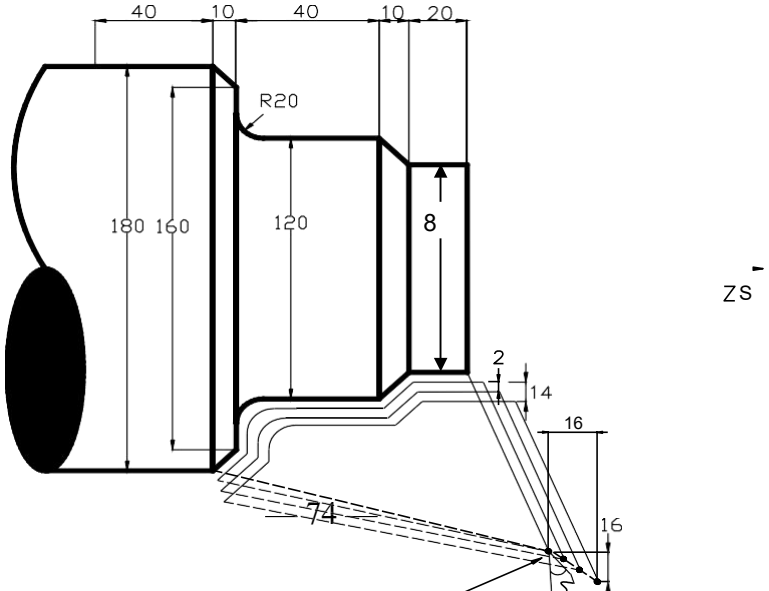
1)Δi < 0 Δk>0, △u < 0 Δw>0; 2)Δi>0 Δk>0, △u>0 Δw>0;



3)Δi<0 Δk <0, Δu<0 Δw<0; 4)Δi>0 Δk <0, △u>0 Δw<0;

Figure 3-27

Example: Figure 3-28



Procedure:

```
O0006;
G99 G00 X200 Z10 M03 S500;        (specify each rotation feed, position starting point, start spindle)
G73 U1.0 W1.0 R3 ;                  (X-axis retreated 2mm,Z-axis retreated 1mm)
G73 P14 Q19 U0.5 W0.3 F0.3 ;      (coarse car, 0.5mm for X axis and 0.3mm for precision car
                                    allowance for Z axis)
N14 G00 X80 Z0 ;
G01 W-20 F0.15 S600,
X120 W-10;                          } finishing shape segments
W-20;
G02 X160 W-20 R20 ;
N19 G01 X180 W-10;
G70 P14 Q19                                  (Finishing)
M30;
```

Finishing cycle G70

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

Command function: the tool performs finishing machining from the starting point along the workpiece finishing trajectory given by the ns ~ nf program segment. After rough machining in G71, G72 or G73, finish turning with G70 instruction to finish cutting the finishing allowance in a single time. At the end of the G70 cycle, the tool returns to the starting point and executes the next segment after the G70 segment.

where: ns: program segment number of the first program segment of the fine vehicle trajectory;

nf: program segment number of the last program segment of the fine vehicle trajectory;

The G70 instruction trajectory is determined by the programming trajectory of the program segment between ns and nf. The relative positions of ns and nf in the G70 ~ G73 program segment are as follows:

```
        .....
G71/G72/G73.....;
N_ (ns) .....
  .....
  • F
  • S                    finishing route program segment group
  •
     N __(nf) .....
          ...
G70 P(ns)  Q(nf);
    ...
```

Instruction Description:

- G70 must be written after ns ~ nf program segment. If it is written before the ns ~ nf program segment, the system will automatically search for the ns ~ nf program segment and execute it. After the execution is completed, the next 1 program of the G70 program segment will be executed in sequence, thus causing repeated execution of the ns ~ nf program segment.
- When performing the G70 finishing cycle, the F, S, and T instructions in the ns ~ nf program section are valid.
- The G96, G97, G98, G99, G40, G41, G42 commands are valid for the G70 finishing cycle.
- During the execution of the G70 command, it is possible to stop the automatic operation and move manually, but to perform the G70 cycle again, you must return to the position before the manual movement. If you do not return to continue execution, the following run track will be misaligned.
- Execute the operation of feed hold and single program segment, and pause the program after running the end of the current track.

- The G70 instruction cannot be executed in the input mode, otherwise an alarm will be generated.
- When the compound loop instruction needs to be used multiple times in the same program, ns ~ nf does not allow the same program segment number.

Axial grooving multiple cycles G74

Instruction format: G74 R (e);

G74 X(U) Z(W) P (Δi) Q (Δk) R (Δd) F_;

Instruction meaning: radial (X-axis) feed cycle compound axial intermittent cutting cycle: from the starting point axial (Z-axis) feed, retreat, re-feed... until the cutting to the same position as the Z axis coordinate of the cutting end point, then radial retreat, axial retreat to the same position as the starting point Z axis coordinate to complete an axial cutting cycle; after radial re-feed, the next axial cutting cycle is performed; after cutting to the cutting end point, return to the starting point (the starting point and the end point of G74 are the same), and the axial grooving compound cycle is completed. The radial feed and axial feed directions of G74 are determined by the relative positions of the cutting end points X(U), Z(W) and the starting point. This instruction is used to machine an annular groove or a central deep hole on the end face of the workpiece, and the axial intermittent cutting plays the role of chip breaking and chip removal in time.

Related definitions:

Axial cutting cycle starting point: the position where each axial cutting cycle starts axial feed, expressed as A_n (n = 1,2,3...), the Z axis coordinate of A_n is the same as the starting point A, and the difference between the X axis coordinate of A_n and A_{n-1} is Δ I. The starting point A of the first axial cutting cycle is $_1$ the same as the starting point A, and the x-axis coordinate of the starting point of the last axial cutting cycle (denoted as A_f) is the same as the cutting end point.

End point of axial feed: the end point of axial feed for each axial cutting cycle, expressed as B_n(n=1,2,3……), B_n the Z-axis coordinate is the same as the cutting end point, B_n the X-axis coordinates of a_n same, the end of the last axial feed (denoted as B_f) Same point as the cutting end point;

Radial retract end point: after each axial cutting cycle reaches the axial feed end point, radial retract (retract amount is Δd), expressed C_n(n=1,2,3……),C_n the Z-axis coordinate is the same as the cutting end point, C_n with a_n the difference between the X-axis coordinates is Δd;

Axial cutting cycle end point: the end position of axial retreating from the end of radial retreating, expressed as D_n (n = 1,2,3...), the Z axis coordinate of D_n is the same as the starting point, and the X axis coordinate of D_n is the same as C_n (the difference between the X axis coordinate of A_n is Δd);

Cutting end point: X(U) Z(W) specified position, last axial feed end point B_f.

R(e): Each axial direction (Z axis) axial retract amount after cutting, value range 0~99.999 (unit: mm), no sign.

R(e) executes the command value remains valid, and the basic parameter 141 value No. is changed to e× 1000 (unit: 0.001mm). When R(e) is not input, the value of the basic parameter 141 is used as the axial retrane number.

X: X-axis absolute coordinate value of the cutting end point B_f (unit: mm).

U: the X-axis absolute coordinate difference between the cutting end point B_f and the starting point A (unit: mm).

Z: the absolute coordinate value of the Z axis of the cutting end point B_f (unit: mm).

W: the difference between the Z-axis absolute coordinates of the cutting end point B_f and the starting point A (unit: mm).

P (Δi): The radial (X-axis) amount of cut in a single axial cutting cycle, with a value range of 1~9999999 (unit: mm, diameter value), without sign.

Q: axial (Z axis) (Δk): axial (Z axis) cutting, Z -axis intermittent feed into the amount of the tool, the range of 1~9999999 (unit: mm), no sign.

R (Δd): cutting to the axial cutting end point, the radial (X axis) of the withdrawal amount, the range of 0~99.999 (unit: mm, diameter value), no sign, omit R (Δd), the system default axial cutting end point, the radial (X axis) of the withdrawal amount is 0.

When the instruction words X(U) and P(, Δi) are omitted, the default is to retreated in

the positive direction.

Instruction execution process: as shown in Figure 3-29.

① from the axial cutting cycle starting point A n $_n$ axial (Z axis) cutting feed △k, cutting end point Z axis coordinate is less than the starting point Z axis coordinate, to the Z axis negative feed, otherwise to the Z axis positive feed;

② Axial (Z axis) quickly move the retreating e, and the retreating direction is opposite to the feed direction of ①;

③ If the Z-axis cuts the feed again (Δk + e) and the feed end point is still between the starting point A $_n$ of the axial cutting cycle and the end point B $_n$ of the axial feed, the Z-axis cuts the feed again (Δk + e) and then executes ②; If the Z-axis cuts the feed again (Δk + e), the feed end point reaches point B $_n$ or is not between A $_n$ and B $_n$, the Z axis cutting and feeding to point B $_n$, and then execute ○ 4;

④ Radial (X-axis) quickly move the retreating △d (diameter value) to C point $_n$. When the X-axis coordinate of point B $_f$ (cutting end point) is less than the X-axis coordinate of point A (starting point), retreating to the X-axis is positive, otherwise retreating to the X-axis is negative. ;

⑤ Axial (Z-axis) quickly move the rewind to point Dn, and the nth axial cutting cycle ends. If it is not the last axial cutting cycle, execute ⑥; if it is the last axial cutting cycle, execute ⑦;

⑥ Radial (X-axis) quickly move the feed, and the feed direction is opposite to the retreating direction. If the X-axis feed (△d + △i)(diameter value) is followed, the feed end point is still between point A and point A $_f$ (the last axial cutting cycle start point), the X-axis quickly moves the feed (△d + △i)(radius value), I .e. Dn→A $_{n+1}$, and then execute ① (start the next axial cutting cycle); if X after the shaft feed (△d + △i) (diameter value), the feed end point reaches A. $_f$ point or not in Dn and A $_f$ between points, X-axis fast move to A $_f$ point, and then execute ① to start the last axial cutting cycle;

⑦ X axis fast movement returns to the starting point of A,G74 instruction execution ends.

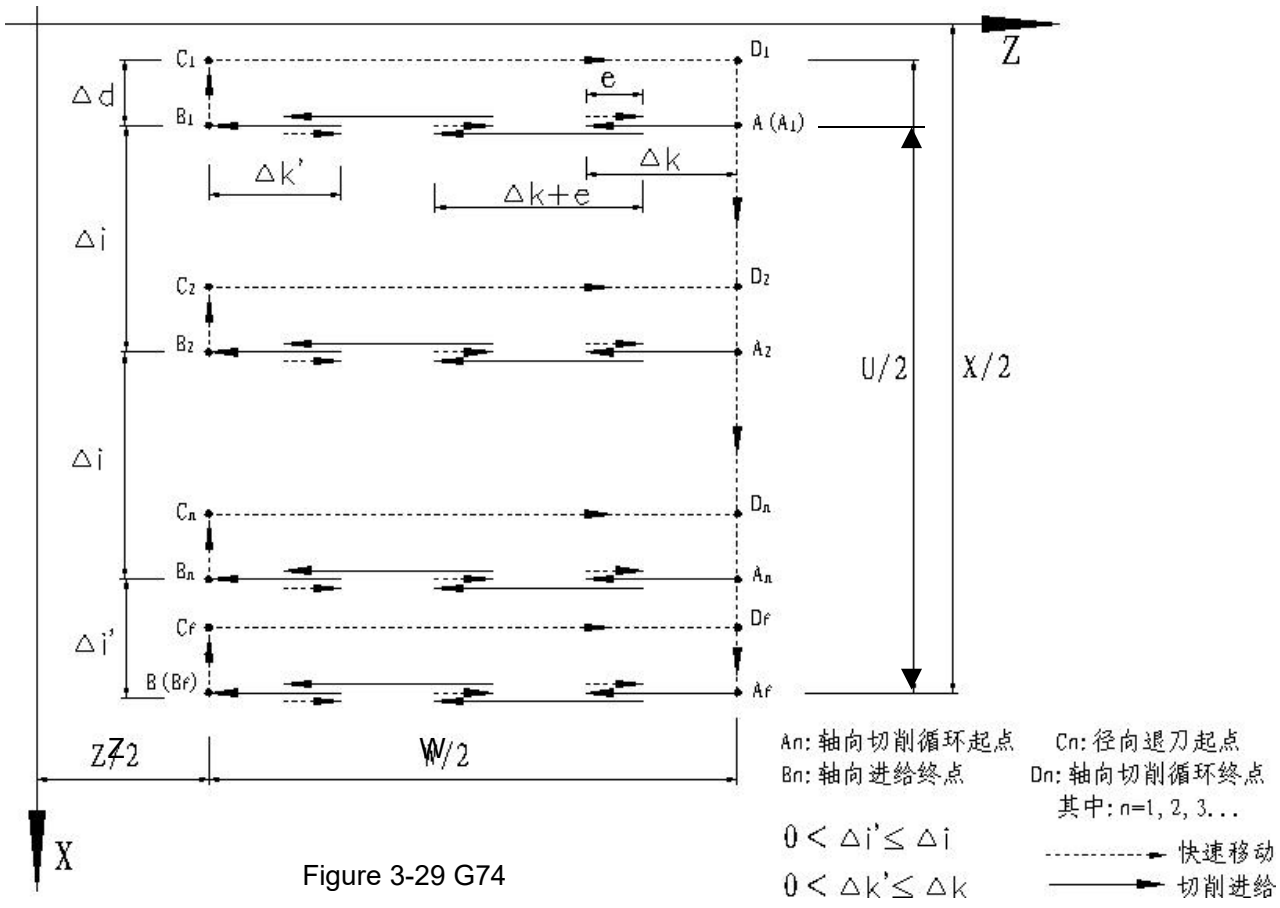


Figure 3-29 G74

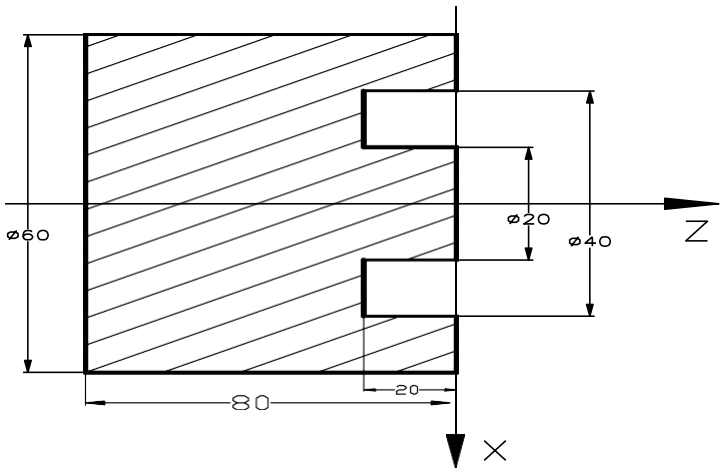
Instruction Description:

● Loop action is composed of Z(W) and Q (Δk) if only" G74 R(e); "Program segment, cycle

action is not performed;

- Δd and e are specified with the same address R, the difference is based on the program segment in the presence of Z(W) and Q (Δk) instruction words;
- During the execution of the G74 command, you can stop the automatic operation and move manually, but to perform the G74 cycle again, you must return to the position before the manual movement. If you do not return to continue execution, the following run track will be misaligned.
- Execute the operation of feed hold and single program segment, and pause the program after running the end of the current track.
- When performing blind hole cutting, the command word R (Δd) must be omitted because there is no withdrawal distance from the cutting to the end of the axial cutting.

example: Figure 3-30, the knife width value is 4mm, the left knife tip is against the knife.



Procedure:

```
O0007;
G0 X40 Z5 M3 S500;          (Start the spindle and position it to the machining starting point)
G74 R0.5 ;                  (Processing cycle)
G74 X28 Z-20 P3 Q5 F50;     (Z axis feed 5mm each time, retreated 0.5mm, feed to the end
                            Z-20,return quickly To the starting point Z5,X-axis feed 3mm, cycle the above steps to continue running)
M30;                        (End of program)
```

Radial grooving multiple cycles G75

Instruction format: G75 R (e);

G75 X(U) Z(W) P (Δi) Q (Δk) R (Δd) F;

Instruction meaning: axial (Z-axis) feed cycle compound radial intermittent cutting cycle: from the starting point radial (X-axis) feed, retreat, re-feed... until the cutting to the same position as the X axis coordinate of the cutting end point, then axial retreat, radial retreat to the same position as the starting point X axis coordinate, complete a radial cutting cycle; after axial re-feed, the next radial cutting cycle is performed; after cutting to the cutting end point, return to the starting point (the starting point and the end point of G75 are the same), and the radial grooving compound cycle is completed. The axial feed and radial feed directions of G75 are determined by the relative position of the cutting end point X(U)Z(W) and the starting point. This instruction is used to process radial annular grooves or cylindrical surfaces, and radial intermittent cutting plays the role of chip breaking and chip

removal in time.

Related definitions:

Radial cutting cycle starting point: each radial cutting cycle starts radial feed position, expressed as A_n (n = 1,2,3...),A_n x axis coordinates and starting point a same, A_n and A n_{-1} of the z axis coordinates of the difference is k. The first radial cutting cycle starting point A is $_1$ the same point as starting point A, and the Z-axis coordinate of the last radial cutting cycle starting point (denoted as A_f) is the same as the cutting end point.

Radial feed end point: the end point position of radial feed in each radial cutting cycle, expressed as B_n (n = 1,2,3...), the X-axis coordinate of B_n is the same as the cutting end point, and the Z-axis coordinate of B_n is the same as A_n, the end point of the last radial feed (denoted as B_f) is the same as the cutting end point;

Axial retreating end point: after each radial cutting cycle reaches the radial feed end point, the axial retreating (the retreating amount is Δd), expressed C $_n$(n=1,2,3……),C_n the X-axis coordinate is the same as the cutting end point, C_n the difference with the An Z-axis coordinate is Δd;

Radial cutting cycle end point: from the end point of axial retreating to the end position of radial retreating, expressed as D_n (n = 1,2,3...), the X-axis coordinate of D_n is the same as the starting point, and the Z-axis coordinate of D_n is the same as C_n (the difference between Z-axis coordinate and A_n is Δd);

Cutting end point: X(U) Z(W) specified position, last radial feed end point B_{-f}.

R (e): the radial retreating amount after each radial (X-axis) feed, with the value range of 0~99.999 (unit: mm), radius value, unsigned. The command value remains valid after R(e) is executed, and the value of the basic parameter 141 number is changed to e× 1000 (unit: 0.001mm). When R(e) is not input, the value of the number of the basic parameter 141 is used as the radial retreating amount.

X: X-axis absolute coordinate value of cutting end point B_f (unit: mm)

U: the X-axis absolute coordinate difference between the cutting end point B_f and the starting point A (unit: mm).

Z: the absolute coordinate value of the Z axis of the cutting end point B_f (unit: mm).

W: the difference between the Z-axis absolute coordinates of the cutting end point B_f and the starting point A (unit: mm).

P(Δi) (: radial (X-axis) feed, X -axis feed intermittent feed amount, the value range is 1~9999999 (unit: mm, diameter value), no sign.

(Δk) Q: single radial cutting cycle of the axial (Z axis), the range of 1~9999999 (unit: mm), no sign.

R(Δd): After cutting to the radial cutting end point, the axial direction (Z axis) regression, value range 0~99.999 (unit: mm), no sign.

Omit R (d), the system default radial cutting end point, the axial (Z -axis) of the withdrawal amount is 0. Z(W) and Q ((Δk) are omitted, and the knife is retracted in the positive direction by default.

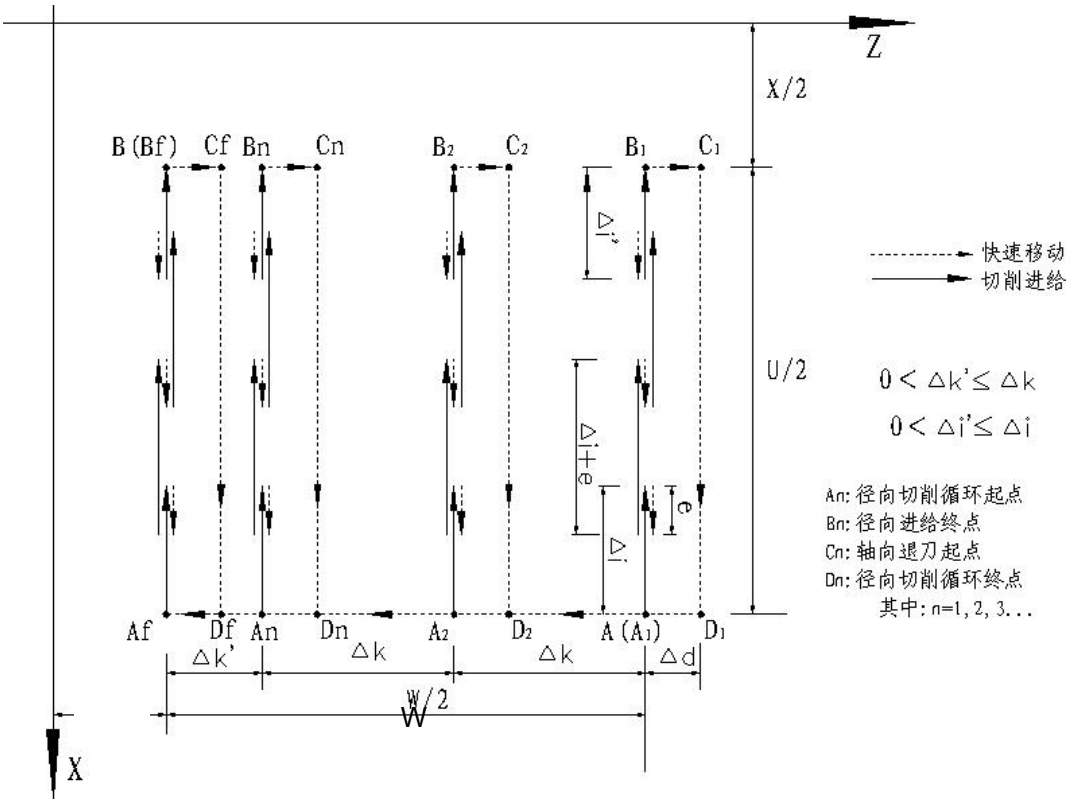


Figure 3-31 G75 trajectory diagra

instruction execution process: figure 3-31

① from the radial cutting cycle starting point A n radial radial (X axis) cutting feed △i, cutting end point X axis coordinates less than the starting point X axis coordinates, to the X axis negative feed, otherwise to the X axis positive feed;

② Radial (X-axis) quickly move the retreating e, and the retreating direction is opposite to the feed direction of ①;

③ If the X-axis cuts the feed again (Δi + e) and the feed end point is still between the starting point A n of the radial cutting cycle and the end point B n of the radial feed, the X-axis cuts the feed again (Δi + e) and then executes ②; If the X-axis cuts the feed again (Δi + e), the feed end point reaches point B n or is not between A n and B n, the X-axis cutting feed to point B n, and then execute ○ 4;

④ Axial (Z axis) quickly move the retreating △d to C point n. When the Z axis coordinate of point B f (cutting end point) is less than the point (starting point) Z axis coordinate, retreating to the Z axis positively, otherwise retreating to the Z axis negatively;

⑤ Radial (X-axis) quickly move the rewind to point Dn, and the nth radial cutting cycle ends. If it is not the last radial cutting cycle, execute ⑥; if it is the last radial cutting cycle, execute ⑦;

⑥ Axial (Z-axis) fast moving feed, feed direction and ④ retreating direction is opposite. If the Z-axis feed (△d + △k), the feed end point is still at point A and A point f (the last radial cutting cycle starting point), the Z-axis rapid movement feed (△d + △k), namely:
Dn→A n+1, and then execute ① (start the next radial cutting cycle); if Z after the shaft feed (△d + △k), the feed end point reaches A. f point or not in Dn and A f between points, the Z axis is fast move to A f point, and then execute ① to start the last radial cutting cycle ring;

⑦ The rapid movement of the Z axis returns to the starting point, A, and the execution of the G75 instruction ends.

Instruction Description:

- Loop action is composed of X(U) and P (Δi) if only" G75 R(e); "Program segment, cycle action is not performed;
- Δd and e are specified with the same address R, the difference is based on the program segment in the presence of X(U) and P (Δi) instruction words;

- During the execution of the G75 command, the automatic operation can be stopped and moved manually, but to perform the G75 cycle again, you must return to the position before the manual movement. If it is executed again without returning, the following running track will be misaligned;
- Execute the operation of feed hold and single program segment, and pause the program after running the end of the current track.
- When performing the grooving cycle, the command word R (Δd) must be omitted because there is no withdrawal distance from the cutting to the radial cutting end point.

Example: Figure 3-32, set the knife width value to 4mm, and set the knife with the left knife tip.

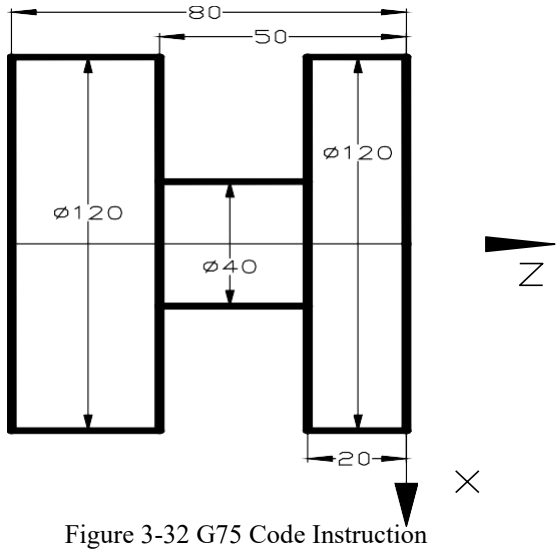


Figure 3-32 G75 Code Instruction

Procedure:

O0008;

G00 X150 Z50 M3 S500; (start the spindle and set the speed to 500)

G0 X125 Z-20; (Positioning to the machining starting point)

G75 R0.5 F150; (machining cycle)

G75 X40 Z-46 P6 Q3; (X axis feed 6mm each time, retreated 0.5mm, feed to the end point (X40),quickly return to the starting point (X125), feed 3mm into the Z axis, and continue to run in the above steps)

G0 X150 Z50; (Return to the starting point of processing)

M30; (End of program)

Thread cutting instructions

With a variety of thread cutting functions, it can process inch/metric single-head, multi-head, variable pitch thread and tapping cycle. The length and angle of thread tail can be variable. Multiple cycle thread cutting can be cut unilaterally, protecting the tool and improving the surface finish. Thread functions include: continuous thread cutting instruction G32, variable pitch thread cutting instruction G34, tapping cycle cutting instruction G33, thread cycle cutting instruction G92, and thread multiple cycle cutting instruction G76.

The machine tool using thread cutting function must be equipped with a spindle encoder. The number of spindle encoder lines is set by the transmission parameter No. 146 parameter. When cutting threads, the system will move the X axis or Z axis and start thread processing only after receiving one rotation signal from the spindle encoder. Therefore, as long as the spindle speed is not changed, the same thread can be cut multiple times by rough turning and finish turning.

It has a variety of thread cutting functions that can be used to process threads without an exit groove. However, due to the acceleration and deceleration process in the X-axis and Z-axis at the beginning and end of thread cutting, the pitch error at this time is large, so it is still necessary to leave the thread lead-in length and the distance between the actual thread start and the end.

Under the condition that the thread pitch is determined, the movement speed of the X-axis and the Z-axis during thread cutting is determined by the spindle speed and has nothing to do with the cutting feed rate. When the spindle speed changes, due to the X-axis, Z-axis acceleration and deceleration will make the pitch error, therefore, do not adjust the spindle speed during thread cutting, and do not stop the spindle, the spindle will cause damage to the tool and workpiece.

Equal pitch thread cutting instruction G32

instruction format: **G32 X(U)_ Z(W)_ F(I)_ J_ K_ Q_**

Instruction function: the movement track of the tool is a straight line from the starting point to the ending point, as shown in fig. 3-33; The coordinate axis with larger displacement from the starting point to the ending point (X axis is based on radius value) is called the long axis, and the other coordinate axis is called the short axis. During the movement process, the long axis moves by one lead every 1 rotation of the main axis, and the short axis and the long axis are linearly interpolated. When the cutting tool cuts the workpiece, to achieve equal pitch thread processing. F and I command words are used to give the pitch of metric and imperial threads respectively. G32 command can be executed to process straight threads, taper threads and end threads of metric or imperial pitch and continuous multi-segment thread processing:

Instruction description: G32 is the modal G instruction;

The pitch of the thread refers to the displacement of the long axis of the 1 rotation of the spindle (the displacement of the X axis is based on the radius value); When the X coordinate values of the starting point and the end point are the same (no X or U is entered), straight thread cutting is performed;

When the Z coordinate values of the start and end points are the same (Z or W is not entered), the end thread cutting is performed; when the X and Z coordinate values of the start and end points are not the same, the taper thread cutting is performed.

F: Metric thread pitch, which is the movement of the long axis of the 1 rotation of the spindle, with a value range of 0.0001~500mm. The F command value remains valid after execution,

Until the F command word for the given thread pitch is executed again.

I: Number of threads per inch, which is the number of threads on a length of 1 inch (25.4mm) in the direction of the long axis, which can also be understood as the movement of the long axis by 1 inch. (25.4mm) when the spindle rotation number of turns. The value range is 0.06~25400 teeth/inch, and the I command value will remain valid after execution until it is executed again.

Line I command word for a given thread pitch.

J: The amount of movement in the minor axis direction when the thread is untailing (untailing amount), the value range is -9999.999~9999.999 (unit: mm), with positive and negative directions;If the minor axis is the X axis, this value is the radius specified; the J value is the modal parameter.

K: The length of the thread in the long axis direction when it is untailed. The value range is 0~9999.999 (unit: mm). If the long axis is the X axis, the value is the radius

Specifies; without direction; the value of K is a modal parameter.

Q: Starting angle, refers to the offset angle between the signal of one revolution of the spindle and the starting point of thread cutting. The value range is 0~360000 (unit: 0.001 degrees).

The Q value is a non-modal parameter and must be specified for each use. If it is not specified, it is considered to be 0 degrees.

Q Usage Rules:

1. If Q is not specified, the default starting angle is 0 degrees;
2. For continuous thread cutting, except for the Q in the first section, the Q specified in the following thread cutting section is invalid, and even if Q is defined, it is ignored;
3. The total number of multi-start threads formed by indexing defined by the starting angle shall not exceed the number of 65535 threads.
4. The unit of Q is 0.001 degrees. If it is offset from the signal of one revolution of the spindle by 180 degrees, Q180000 shall be input in the program. If Q180 is input or Q180.0 are considered to be 0.18 degrees.

How to judge the long and short axes: Figure 3-33.

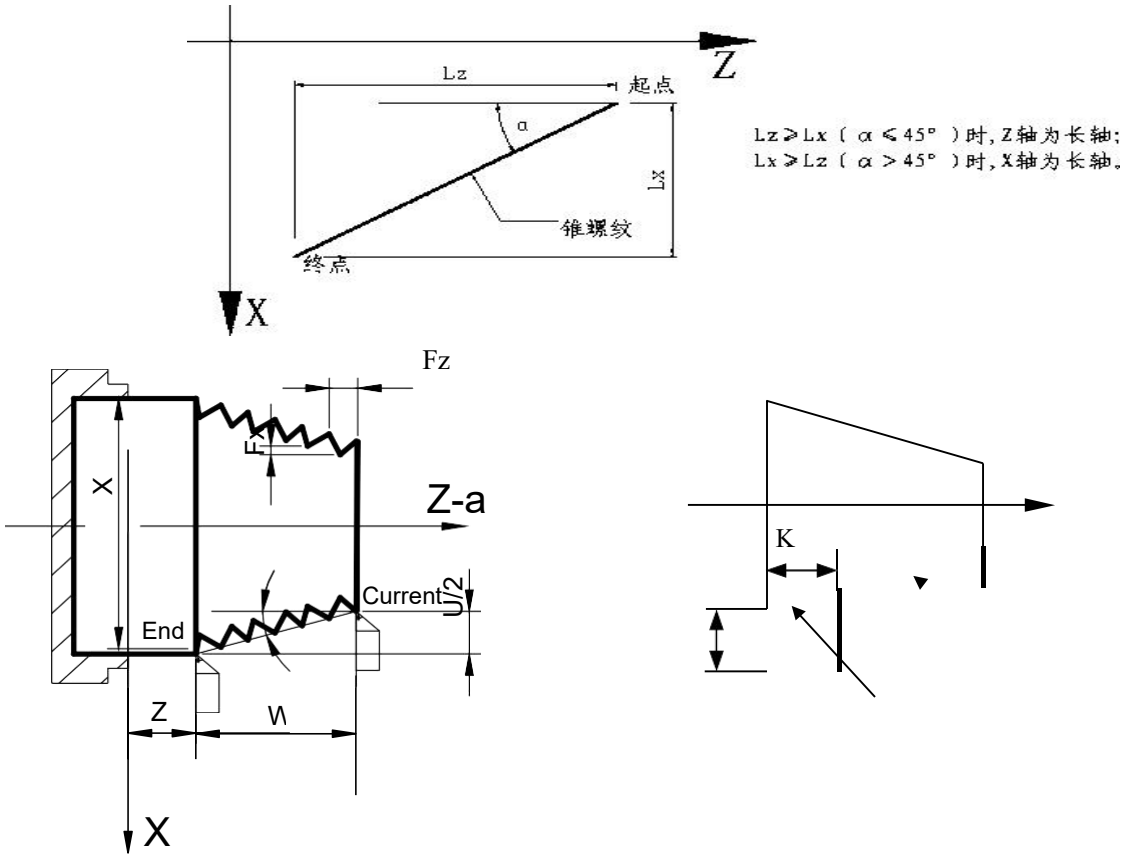


Figure 3-33 G32 trajectory

Tool path

G32 Programming End

Note:

- J and K are modal instructions. When J and K are omitted from the 1 program segment during continuous thread cutting, the tail is untailed according to the previous J and K values, and J and K modes are canceled when non-thread cutting instructions are executed;
- When J or J and K are omitted, there is no tail withdrawal; when K is omitted, press K = J to tail withdrawal;
- When J = 0 or J = 0, K = 0, there is no backoff;
- When J≠ 0 and K = 0, press K = J to back off;
- When J = 0 and K≠ 0, there is no tail-out;
- The current program section is thread cutting, and the next 1 program section is also thread cutting. When the next 1 program section starts cutting, the one-turn signal of the spindle position encoder is not detected, and thread processing is directly started. This function can realize continuous thread processing.
- After performing the feed hold operation, the system displays "pause" and the thread cutting does not stop until the current program section is completed. For continuous thread processing, the thread cutting program section is executed before the movement is stopped, and the program operation is suspen
- In single-segment operation, the movement is stopped after the current program segment is executed, and the movement is stopped only after the thread cutting program segment is executed for continuous thread processing.
- When the system is reset, emergency stop or drive alarm, the thread cutting decelerates and stops.

Example: Thread pitch: 2mm. δ1=3mm,δ2=2mm, total cutting depth 2mm, cut in two times.

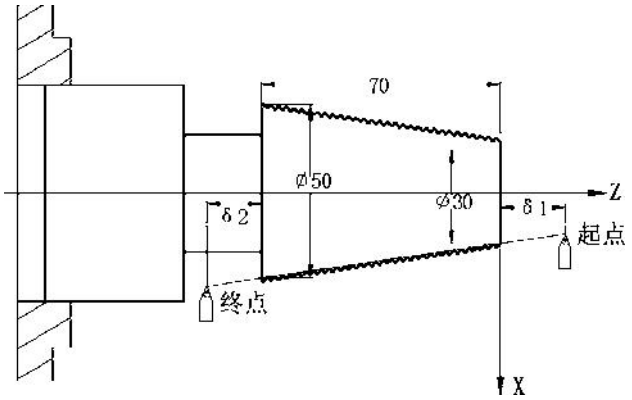


Figure 3-34

procedure:

```
O 0009;
G00 X28 Z3;                    (first cut 1mm)
G32 X51 W-75 F2.0;             (First cutting of conical thread)
G00 X55;                       (tool exit)
W75;                           (Z-axis return to starting point)
X27;                           (0.5mm second re-feed)
G32 X50 W-75 F2.0;             (Second cutting of conical thread) G00
X55;                           (tool exit)
W75 ;                          (Z-axis return point);
M30
```

Variable Pitch Thread Cutting Command G34

Command format: G34 X(U) __ Z (W) __ F (I) __ J_ K__ R;

command function: the motion trajectory of the tool is a straight line from the starting position of the X and Z axes to the end position specified by the program section. The coordinate axis with larger displacement from the starting point to the ending point (X axis is based on radius value) is called the long axis, and the other coordinate axis is called the short axis. During the movement, the long axis moves 1 lead per revolution, and the pitch of the spindle moving 1 revolution is continuously increasing or decreasing the specified value, forming a variable pitch spiral cutting groove on the surface of the workpiece to realize the processing of variable pitch thread. When cutting, you can set the amount of retreated.

F and I command words are used to give the pitch of metric and imperial threads respectively. G34 command can be executed to process straight threads, taper threads and end threads of metric or imperial variable pitch.

Instruction description: G34 is the modal G instruction;

The meanings of X(U), Z(W), J and K are consistent with G32;

F: Metric thread of the first pitch starting from the starting point coordinate value, with the value range of 0.001~500mm;

I: English threads of the first pitch starting from the starting coordinate value, ranging from 0.06 to 25400 teeth/inch;

R: increment value or decrement value of the spindle pitch per revolution, R = F1-F2,R with direction; When F1>F2, when R is negative, the pitch decreases; When F1<F2, the pitch increases when R is positive (as shown in Fig. 3-35);

R value range: ± 0.001 ~ ± 500.000mm/ pitch (metric thread);

± 0.060 to ± 25400 teeth per inch (imperial thread).

When the R value exceeds the above range value and the pitch exceeds the allowable value or the pitch has a negative value due to the increase/decrease of R.

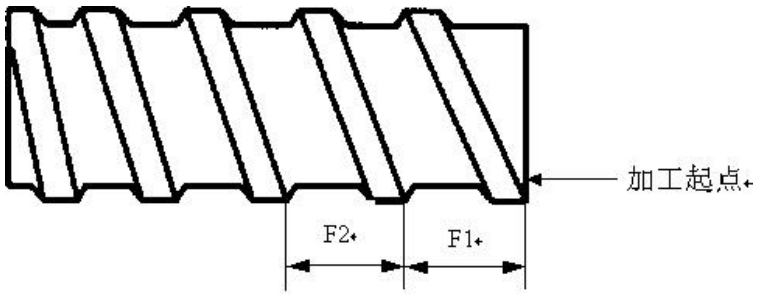


图 3-35 变螺距螺纹示图

Note:

- Precautions are the same as for G32 thread cutting.

Example: The first pitch of the starting point is 4mm, and the increment value of the pitch per revolution of the spindle is 0.2.

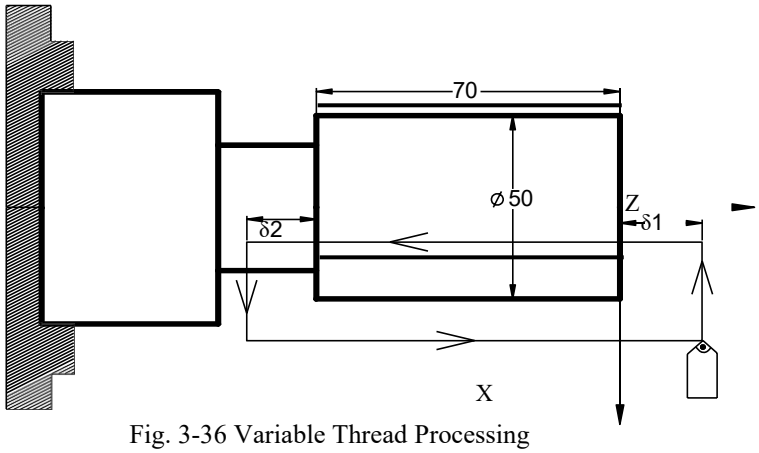


Fig. 3-36 Variable Thread Processing

When cutting with the G34 cycle multiple times, use macro variables to simplify programming. Values: δ1=4mm,δ2=4mm, total cutting depth 4mm, total cutting cycle 15 times; The first feed is 0.8mm, each decreasing cutting amount is 0.2mm, and the minimum feed amount is 0.2mm.

Procedure: O0010;

```
G00 X60 Z4 M03 S500;
G65 H01 P#202 Q800;                    First feed: assignment #202=0.8mm
G65 H01 P#203 Q0;                      cycle count: assignment #203=0
N10 G65 H02 P#204 Q#203 R1;            cycle count start:#204 =#203+1
G65 H01 P#203 Q#204;                                  #203 =#204

G65 H81 P30 Q#204 R15;                 Total number of cutting cycles:#204 =, 15, transfer to
G00 U-10;                              N30 program segment feed to Φ 50
G65 H01 P#200 Q#202;                   Cutting feed:#200 =#202
G00 U-#200;                            feed
G34 W-78 F3.8 J5 K2 R0.2;              Variable Pitch Thread Cutting

G00 U10;                               Withstand

Z4;                                    Z-axis return start point

G65 H03 P#201 Q#200 R200;              Decrease of re-cutting feed:#201 =#200-0.2
G65 H01 P#202 Q#201;                   Re-assignment #202 =#201
G65 H86 P20 Q#202 R200;                Feed judgment: when #202 ≦ 0.2mm, transfer to
G65 H80 P10;                           N20 program segment and transfer to N10 program
                                       segment unconditionally
N20 G65 H01 P#202 R200;                Minimum feed:#202=0.2
                                       transfer to N10 program
G65 H80 P10;                           section unconditionally
N30 M30;
```

Z-axis tapping cycle G33

Command format: G33 Z(W) __ F(I) L;
—

Command function: the movement trajectory of the tool is from the starting point to the end point, and then from the end point back to the starting point. In the process of movement, the spindle moves a pitch every 1 turns of the Z axis, which is always consistent with the pitch of the tap. A spiral groove is formed in the inner hole of the workpiece, and the thread processing of the inner hole can be completed at one time.

Instruction description: G33 is the modal G instruction;

Z(W): When Z or W is not entered, the Z coordinate value of the start point and the end point are the same, and thread cutting is not performed;

F: Metric thread pitch, value range 0.001~500mm;

I: per inch of thread number, the value range is 0.06~25400 teeth/;

L: the number of multiple thread heads, the value range is 1~99, can not omit the L value, no alarm.

Cyclic process:

①Z-axis feed tapping (spindle must be specified before G33 instruction);

② After reaching the specified Z-axis coordinate end point, M05 signal output;

③ After detecting the complete stop of the spindle;

④ Spindle reverse signal output (opposite to the original spindle rotation direction);

⑤ Z -axis retreating to the starting point;

⑥ M05 signal output, spindle stops;

⑦ If it is a multi-head thread, repeat steps ① ~ ⑤.

Example of procedure: Figure 3-37, Thread M10 × 1.5

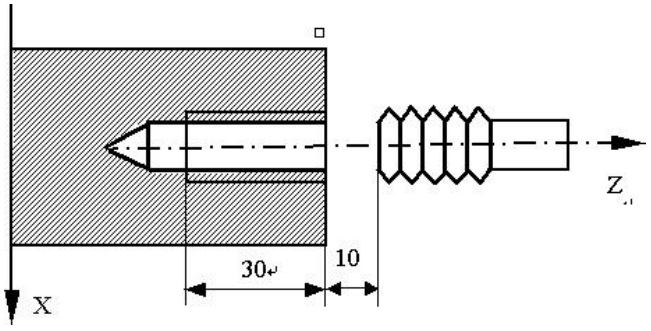


Figure 3-37

Procedure:

```
O0011;
G00 Z90 X0 M03;          starting spindle
G33 Z50 L1 F1.5;            tapping cycle
M03                      restart spindle
G00 X60 Z100;            continue machining
M30
```

Note **1**: Before tapping, the rotation direction of the spindle should be determined according to the rotation direction of the tap. After tapping, the spindle will stop rotating. If you need to continue processing, you need to restart the spindle.

Note **2**: This command is rigid tapping. After the spindle stop signal is valid, the spindle will stop rotating after a certain deceleration time. At this time, the **Z** axis will still feed with the rotation of the spindle until the spindle stops completely. Therefore, the bottom hole position of the thread during actual processing is slightly deeper than the actual programmed position. The specific excess length is determined by the spindle speed during tapping and the spindle brake device.

Note **3**: The moving speed of the **Z** axis during tapping is determined by the spindle speed and pitch, and has nothing to do with the cutting feed rate.

Note **4**: When running in a single program segment or performing a feed hold operation, the system will display "pause", and the tapping cycle will not stop until it returns to the starting point after the tapping is completed.

Note **5**: the system reset, emergency stop or drive alarm, tapping cutting deceleration stop.

Thread cutting cycle G92

Command format: G92 X(U)_ Z(W)_ F_ J_ K_ L ; (Metric straight thread cutting cycle)
G92 X(U)_ Z(W)_ I_ J_ K_ L ; (Imperial straight thread cutting cycle)

G92 X(U)_ Z(W)_ R_ F_ J_ K_ L ; (Metric taper thread cutting cycle)
G92 X(U)_ Z(W)_ R_ I_ J_ K_ L ; (Inch taper thread cutting cycle)

Command function: starting from the cutting starting point, radial (X axis) feed, axial (Z axis or X, Z axis at the same time) cutting, to achieve equal pitch of straight thread, taper thread cutting cycle. Execute G92 instruction, and there is a thread untailing process at the end of thread processing: at a fixed length from the thread cutting end point (called the thread untailing length), while the Z axis continues thread interpolation, the X axis accelerates the exit along the exit direction index or linear (set by parameters). after the Z axis reaches the cutting end point, the X axis then exits at a fast moving speed, as shown in fig. 3-41.

Instruction description: G92 is the modal G instruction;
Cutting Start Point: the starting position of thread interpolation;
Cutting End Point: the ending position of thread interpolation;

X: absolute coordinate of the X axis of the cutting end point, unit: mm;
U: difference between the absolute coordinate of the X axis of the cutting end point and the starting point, unit: mm;
Z: absolute coordinate of the Z axis of the cutting end point, unit: mm;
W: difference between the absolute coordinate of the Z axis of the cutting end point and the starting point, unit: mm;

R: The difference (radius value) between the absolute coordinates of the X axis of the cutting start point and the cutting end point. When the symbols of R and U are inconsistent, it is required

√R｜ ≤ ｜ U/2 ｜, unit: mm;

F: metric thread pitch, the value range is 0.0001~500mm, and the F command value is maintained after execution, and input can be omitted;

I: The number of teeth per inch of I inch thread, the value range is 0.06~25400 teeth/inch, I command value is maintained after execution, and input can be omitted;

J: The amount of movement in the minor axis direction when the thread is untailing (untailing amount), the value range is -9999.999~9999.999 (unit: mm), with positive and negative directions;If the minor axis is the X axis, this value is the radius specified; the J value is the modal parameter.

K: The length of the thread in the long axis direction, ranging from 0 to 9999.999 (unit: mm) ,. Without direction, modal parameters, such as the long axis is the X axis, the value is the radius specified;

L: multi-head thread head number, the value range is: 1~99 modal parameters. (L is omitted by default for single thread)

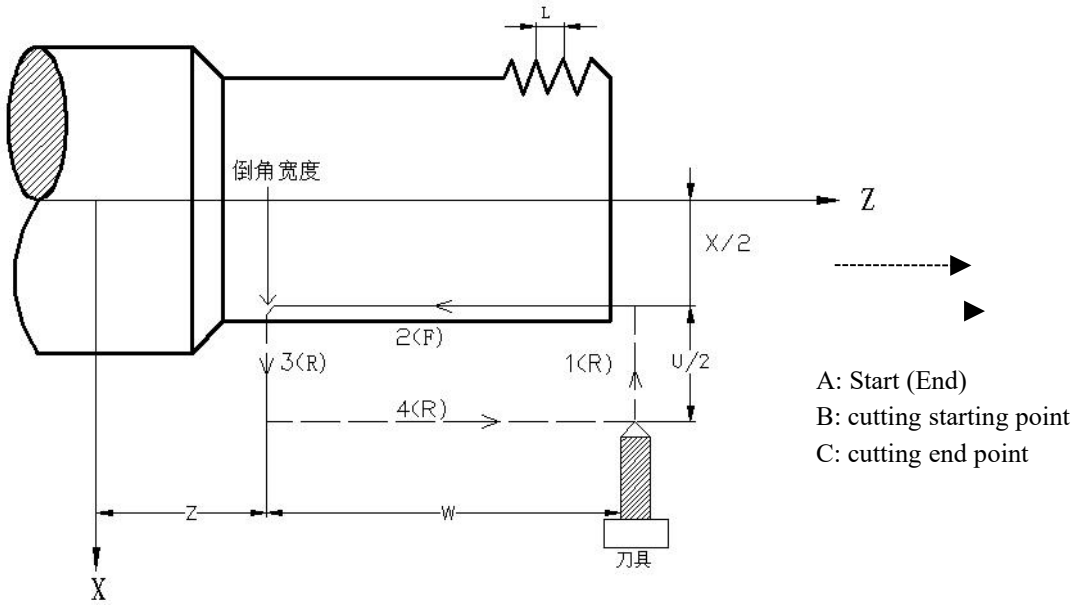


Figure 3-38

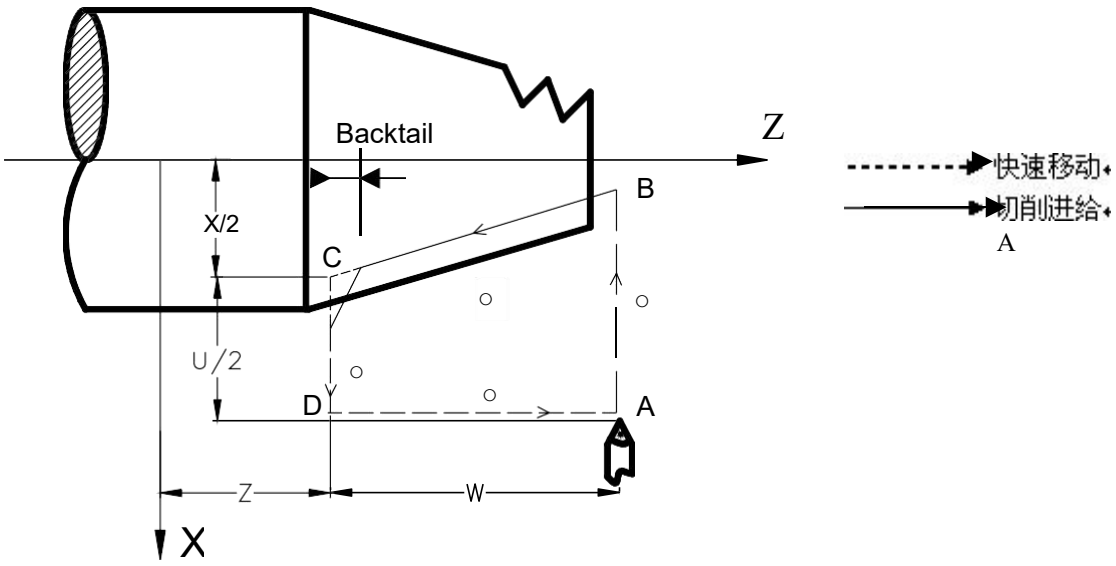


Figure 3-39

The G92 instruction can complete the processing of one thread in multiple feed, but it cannot realize the processing of two continuous threads, nor can it process the end face thread.

The definition of G92 command thread pitch is consistent with G32. Pitch refers to the displacement of the long axis of the spindle 1 rotation (the displacement of the X axis is based on the radius value). Taper thread pitch refers to the spindle rotation 1 the long axis of the displacement (X axis displacement according to the radius value),B and C point Z axis coordinate difference absolute value is greater than the X axis (radius value) coordinate difference of the absolute value, Z axis is the long axis; On the contrary, the X axis is the long axis.

Cycle process: straight thread as shown in Figure 3-38, taper thread as shown in Figure 3-39.

① X axis moves quickly from the starting point to the cutting starting point;

② thread interpolation from cutting start point to cutting end point;

③ The X axis is retreated at a fast moving speed (opposite to the direction of ①), and the X axis absolute coordinate is the same as the starting point;

④ The Z axis moves quickly back to the starting point and the cycle ends.

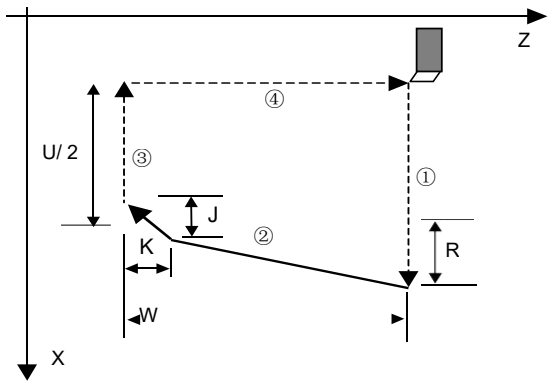
Note:

- When J and K are omitted, the end shall be withdrawn according to the setting value of basic parameter No. 7;
- When J is omitted, the long axis direction is retreated according to K, and the short axis direction is retreated according to the basic parameter No. 70 parameter setting value;
- When K is omitted, press K = J to back out;
- When J = 0 or J = 0, K = 0, there is no backoff;
- When J≠ 0 and K = 0, press K = J to back off;
- When J = 0 and K≠ 0, there is no tail-out;
- After performing the feed hold operation during thread cutting, the system still performs thread cutting, thread cutting is completed, and "pause" is displayed ",

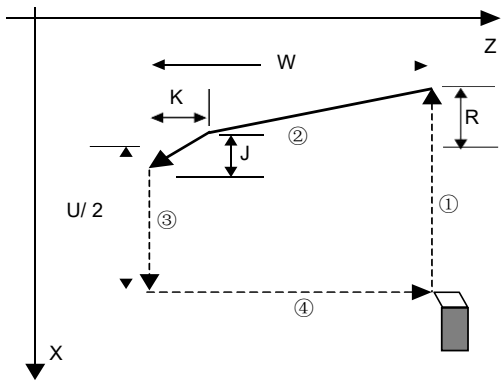
  Program operation is suspended;
- After the single segment operation is executed in the thread cutting process, the operation stops after returning to the starting point (one thread cutting cycle is completed);
- When the system is reset, emergency stop or drive alarm, the thread cutting decelerates and stops.

Command path: U, W and R reflect the relative position of the thread cutting end point and starting point, and the tool path and tail-out direction of U, W and R when the symbols are different, as shown in Figure 3-40:

1) U>0, W<0, R>0



2) U<0, W<0, R<0



3) U>0, W>0, R<0), ｜R｜≤ ｜U/2｜

4) U<0, W>0, R>0, ｜R｜≤｜U/2｜

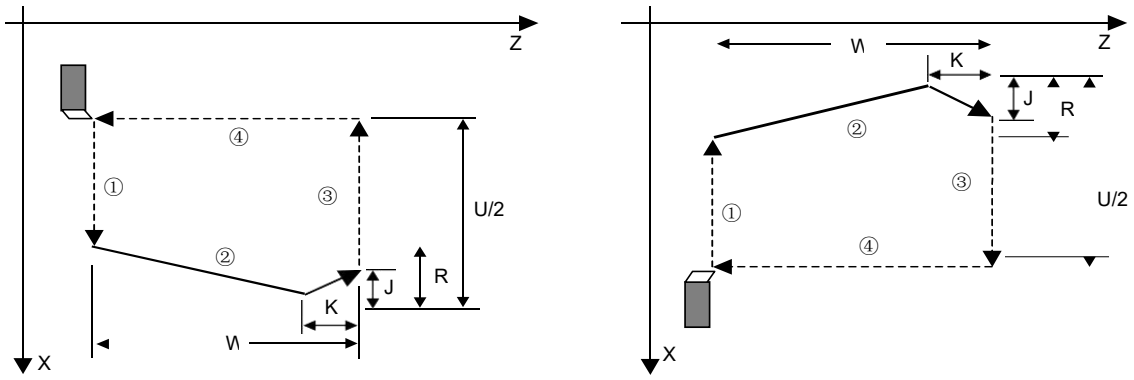


Figure 3-40

Example: Figure 3-41

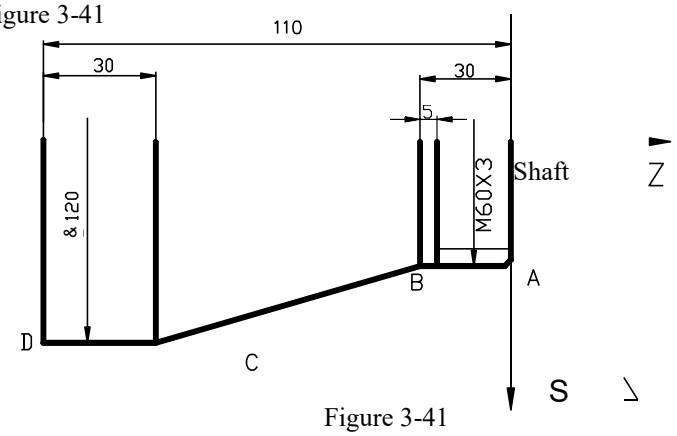


Figure 3-41

Procedure:

```
O0012;
M3 S300 G0 X150 Z50 T0101;          (Thread cutter)
G0 X65 Z5;                          (rapid positioning)
G92 X58.7 Z-28 F3 J3 K1;            (Thread processing, 4-blade cutting, first feed 1.3mm)
X57.7 ;                             (Second feed 1mm)
X57;                                (Third feed 0.7mm)
X56.9;                              (Fourth feed 0.1mm)
M30;
```

Multiple Thread Cutting Cycles G76

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

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

Command function: Through multiple rounds of rough threading and fine threading, the specified thread height (total cutting depth) is processed. If the defined thread angle is not 0 °, the entry point of rough threading gradually moves from the top of the thread to the bottom of the thread, so that the angle between adjacent two threads is the specified thread angle. The G76 instruction can process straight and tapered threads with thread tails, achieving single edge thread cutting with a gradual reduction in blade feed, which is beneficial for protecting the tool and improving thread accuracy. The G76 command cannot process end face threads. The machining trajectory is shown in Figure 3-42.

Related definitions:

Starting point (endpoint): The position of the program segment before and at the end of operation, represented as point A;

Thread end point: The thread cutting end point defined by X (U) Z (W), represented as point D. If there is thread tailing, the long axis direction of the cutting end point is the thread cutting end point, and the short axis direction is the position after tailing.

Thread starting point: The absolute coordinates of the Z-axis are the same as point A, and the difference between the absolute coordinates of the X-axis and the X-axis of point D is i (thread taper and radius values), denoted as point C. If the defined thread angle is not 0°, the cutting cannot reach point C;

Thread cutting depth reference point: The Z-axis absolute coordinate is the same as point A, and the difference between the X-axis absolute coordinate and the C-axis X-axis absolute coordinate is k (the total cutting depth and radius value of the thread), denoted as point B. The thread cutting depth of point B is 0, which is the reference point for the system to calculate each thread cutting depth;

Thread cutting depth: The cutting depth of each thread cutting cycle. The intersection point between the reverse extension line of each thread cutting trajectory and the straight line BC, and the difference (unsigned, radius value) between this point and the absolute coordinate of the X-axis of point B, is the thread cutting depth. The thread cutting depth for each rough turning is $\sqrt{n}$ × △ d, n is the current number of rough turning cycles, and △ d is the thread cutting depth of the first rough turning;

Thread cutting amount: difference between the current thread cutting depth and the previous thread cutting depth: ($\sqrt{n}$ - $\sqrt{n-1}$) × △ d;

End point of tool retraction: The end point position of the radial (X-axis) tool retraction after the end of thread cutting in each coarse and fine thread turning cycle, denoted as point E;

Thread cutting point: The actual starting point of thread cutting in each thread rough turning cycle and precision turning cycle, denoted as Bn point (n is the number of cutting cycles), B1 is the first thread rough turning point, Bf is the last thread rough turning point, and Be is the thread precision turning point.

The displacement of point B_n relative to the X and Z axes of point B follows the formula:

$$tg\frac{a}{2}=\frac{|\text{Z-axis displacement}|}{|\text{X-axis displacement}|}$$ a: **thread angle**

X: Absolute coordinate of the X-axis at the end point of the thread (unit: mm);

U: The difference in absolute coordinates between the end point of the thread and the starting point X-axis (unit: mm);

Z: The absolute coordinate value of the Z-axis at the end point of the thread (unit: mm);

W: The difference in absolute coordinates between the end point of the thread and the starting point Z-axis (unit: mm);

P (m): The number of thread precision turning times is 01-99 (unit: times), and the value of the m command remains valid after execution, and the value of the system basic parameter 142 is modified to m. When m is not entered, the value of system basic parameter 142 is used as the number of precision turns. During thread finishing, the cutting amount of each feed is equal to the cutting amount d of thread finishing divided by the number of finishing turns m.

P (r): Thread tailing length 00-99 (unit: 0.1) × L. L is the thread pitch), the r command value remains valid after execution, and the value of system basic parameter 70 is modified to r. When r is not entered, the value of system basic parameter 70 is used as the thread tail width. The thread unwinding function can achieve thread

processing without a cutter groove. The thread unwinding width defined by the system's basic parameter 70 is valid for G92 and G76 commands;

P (a): The angle between two adjacent threads, with a value range of 00-99, in degrees (°). The value of command a remains valid after execution, and the value of system basic parameter 143 is modified to a. When a is not entered, use the value of system basic parameter 143 as the angle of the thread tooth. The actual angle of the thread is determined by the tool angle, so a should be the same as the tool angle; Q (△dmin): The minimum cutting amount during thread roughing, with a value range of 00-99999 (unit: 0.001mm, unsigned, radius value). When ($\sqrt{n}$ - $\sqrt{n-1}$) ×△ d<△dmin, use △dmin as the cutting amount for this rough turning, that is, the thread cutting depth for this round is ($\sqrt{n-1}$× △ d+△dmin). The setting of Δ dmin is to avoid reducing the amount of rough cutting and increasing the number of rough turns due to the decreasing amount of thread rough cutting. After Q (Δdmin) is executed, the instruction value Δdmin remains valid and the value of system basic parameter 144 is modified to Δdmin (unit: 0.001 mm). When Q (△dmin) is not input, the minimum cutting amount is taken as the value of system basic parameter 144;

R (d): The cutting amount of thread precision turning, with a value range of 00-99.999, (unit: mm, unsigned, radius value). The radius value is equal to the difference between the X-axis absolute coordinates of the thread precision turning entry point Be and the last thread rough turning entry point Bf. After R (d) is executed, the instruction value d remains valid and the value of system basic parameter 145 is modified to d × 1000 (unit: 0.001 mm). When R (d) is not inputted, the value of system basic parameter 145 is used as the thread finishing cutting amount;

R (i): Thread taper, the difference between the absolute coordinates of the X-axis of the thread starting point and the thread ending point, with a value range of -9999999 to 9999999 (unit: mm, radius value). When R (i) is not input, the system processes it as R (i)=0 (straight thread);

P (k): Thread tooth height, total cutting depth of the thread, value range from 1 to 9999999 (unit: 0.001mm, radius value, unsigned). When P (k) is not input, the system alarms;

Q (△ d): The depth of the first thread cutting, ranging from 1 to 9999999 (unit: 0.001mm, radius value, unsigned). When △ d is not input, the system alarms;

F: Metric thread pitch, ranging from 0.0001 to 500 mm;

I: The number of threads per inch for English threads, with a range of 0.06-25400 threads per inch;

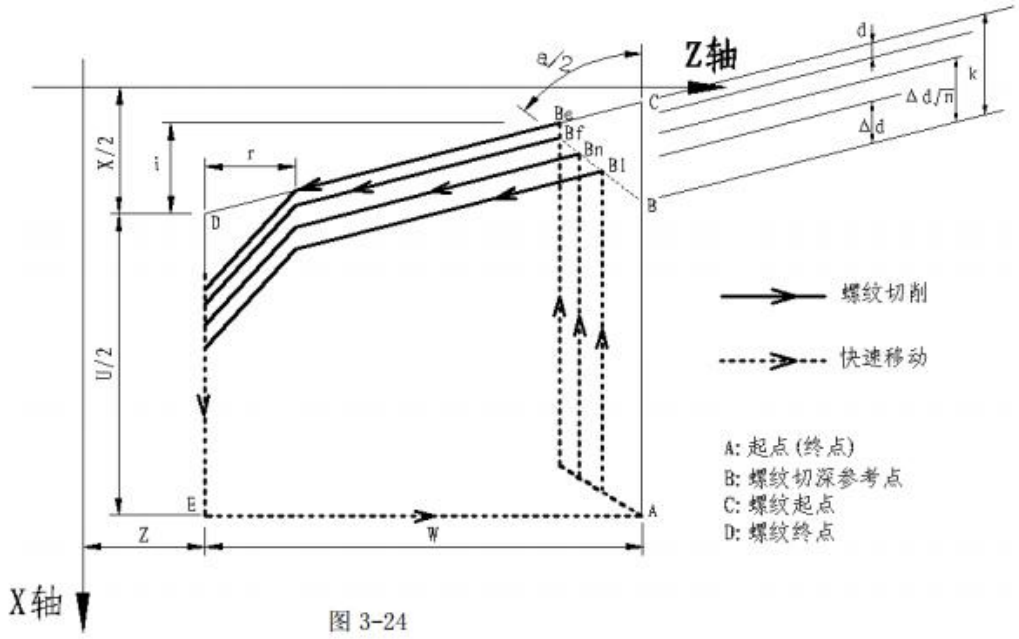


图 3-24

The detailed situation of the entry method is shown in Figure 3-4:

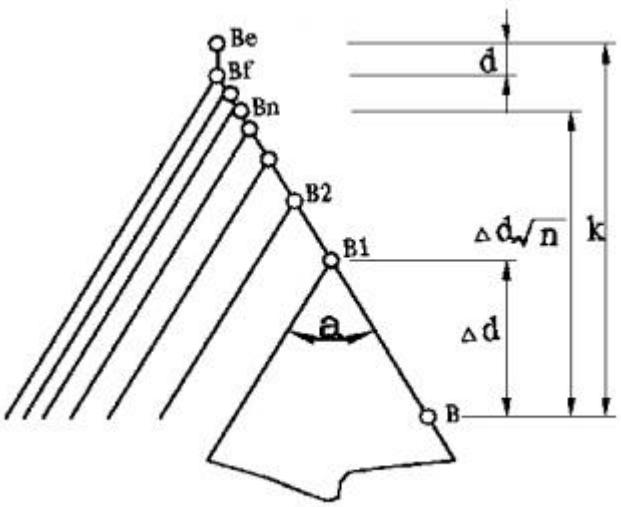


图 3-43

The thread pitch refers to the displacement of the main axis after one revolution of the main shaft (the displacement of the X-axis is based on the radius value). When the absolute value of the Z-axis coordinate difference between point C and point D is greater than the absolute value of the X-axis coordinate difference (the radius value is equal to the absolute value of i), the Z-axis is the main axis; On the contrary, the X-axis is the major axis.

Instruction execution process:

① Quickly move from the starting point to B1, and cut the thread to a depth of △ d. If a=0, only move the X-axis; If a ≠ 0, the X-axis and Z-axis move simultaneously, and the direction of movement is the same as that of A → D;

② Cut the thread parallel to C → D until it intersects with D → E (there is a tailing process when r ≠ 0);

③ Quickly move the X-axis to point E;

④ The Z-axis quickly moves to point A, completing a single rough turning cycle;

⑤ Quickly move the feed again to Bn (n is the number of rough turns), and take the cutting depth ($\sqrt{n}$ × △ d), ($\sqrt{n-1}$ × The larger value in △d+△dmin, if the cutting depth is less than (k-d), proceed to ② for execution; If the cutting depth is greater than or equal to (k-d), feed according to the cutting depth (k-d) to point Bf, and turn ⑥ to perform the last rough thread turning;

⑥ Cut the thread parallel to C → D until it intersects with D → E (there is a tailing process when r ≠ 0);

⑦ Quickly move the X-axis to point E;

⑧ The Z-axis quickly moves to point A, the thread rough turning cycle is completed, and the thread fine turning begins;

⑨ After quickly moving to point Be (with thread cutting depth of k and cutting amount of d), perform thread precision turning, and finally return to point A to complete a thread precision turning cycle;

⑩ If the number of finishing cycles is less than m, turn ⑨ to proceed to the next finishing cycle, and the thread cutting depth remains k and the cutting amount is 0; If the number of precision turning cycles is equal to m, the G76 composite thread processing cycle ends.

Precautions:

●After executing the feed holding operation during thread cutting, the system still performs thread cutting. After thread cutting is completed, "pause" is displayed, and the program operation is paused;

●During the thread cutting process, a single pass sequence operation is executed, and the operation stops after returning to the starting point (one thread cutting cycle action is completed);

●When the system is reset, emergency stop or drive alarm occurs, thread cutting deceleration stops;

●G76 P (m) (r) (a) Q (△dmin) R (d) can omit all or part of the instruction addresses, and the omitted addresses run according to the parameter settings;

●When m, r, and a are inputted with the same instruction address P once, and all m, r, and a are omitted, run according to the set values of parameters NO.57, 19, and 58; When entering 1 or 2 digits for address P, the value is a; When entering 3 or 4 digits for address P, the values are r and a;

●The symbols of U and W determine the direction of A → C → D → E, while the symbol of R (i) determines the direction of C → D. U. The symbol W has four combinations, corresponding to four machining trajectories, as shown in Figure 3-4.

Example: Figure 3-4, thread M68 × 6.

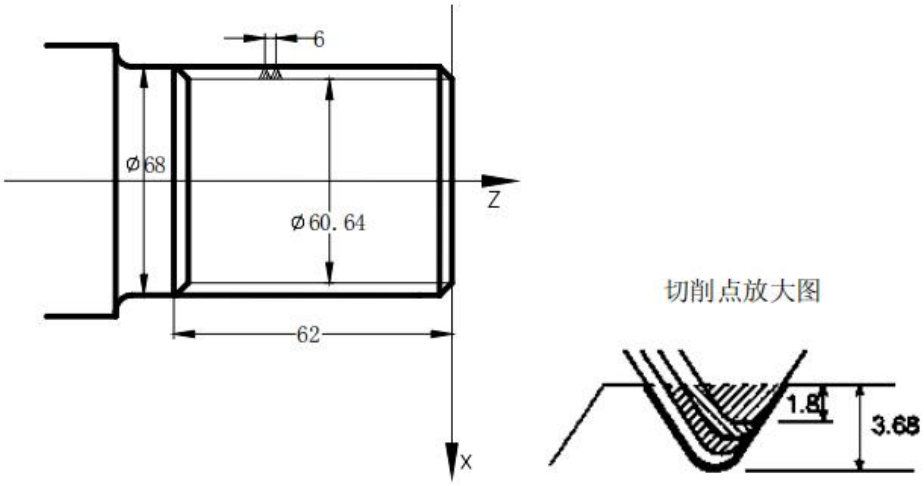


图 3-44

Program:
O0013;
G50 X100 Z50 M3 S300; (Set the workpiece coordinate system to start the spindle and specify the speed)
G00 X80 Z10; (Quickly move to the machining starting point)
G76 P020560 Q0.15 R0.1; (Number of precision machining repetitions 2, chamfer width 0.5mm, tool angle 60°,minimum cutting depth 0.15, precision turning allowance 0.1)
G76 X60.64 Z-62 P3.68 Q1.8 F6; (Thread thread height 3.68, first thread cutting depth 1.8)
G00 X100 Z50; (Return to program starting point)
M30; (End of program)

Constant linear speed control G96, constant speed control G97

Instruction format: G96 S xxxx ;(S0000 ~ S9999, leading zeros can be omitted)

command function: constant line speed control is effective, given cutting line speed (m/min),

cancel constant speed control. G96 is a modal G command. If the current

mode is G96, G96 may not be entered. G96 may not be entered.
Instruction format: G97 S xxxx ;(S0000 ~ S9999, leading zeros can be omitted)

command function: cancel the constant linear speed control, constant speed control is effective,

given the spindle speed (rev/min). G97 is a modal G command. If the

current mode is G97, G97 may not be entered. G97
Command format: G50 S xxxx ;(S0000 ~ S9999, leading zeros can be omitted)

Command function: set the maximum spindle speed limit value (rpm) during constant linear speed control,

and will not use the current position as the program zero point.

G96 and G97 are modal instruction words of the same group, and only one can be valid at the same time. G97 is the initial instruction word, G97 is valid when the system is powered on. When turning a workpiece, the workpiece usually rotates with the spindle axis as the center line,

and the cutting point of the tool cutting the workpiece can be regarded as around the spindle

axis.Line for circular motion, the instantaneous rate of circumferential tangent direction is called cutting speed (usually referred to as line speed). Different materials of the workpiece, different materials of the tool requirements of the line speed is different.
When the spindle speed analog voltage control function is effective, the constant linear speed control function is effective. In constant linear speed control, the spindle speed with the programmed trajectory The absolute value of the X-axis absolute coordinate value changes (ignoring the tool length compensation), the absolute value of the X-axis absolute coordinate value increases, the spindle speed decreases, the X -axis absolute coordinate value decreases, and the spindle speed increases, so that the cutting linear speed is maintained at the S command value. The constant wire speed control function is used to cut the workpiece, which can keep the surface finish of the workpiece with varying diameter.

Linear speed = spindle speed × | X | × π ÷ 1000 (m/min)

spindle speed: rpm

|X|: the absolute value (diameter value) of the X-axis absolute coordinate value, mm

π ≈ 3.14

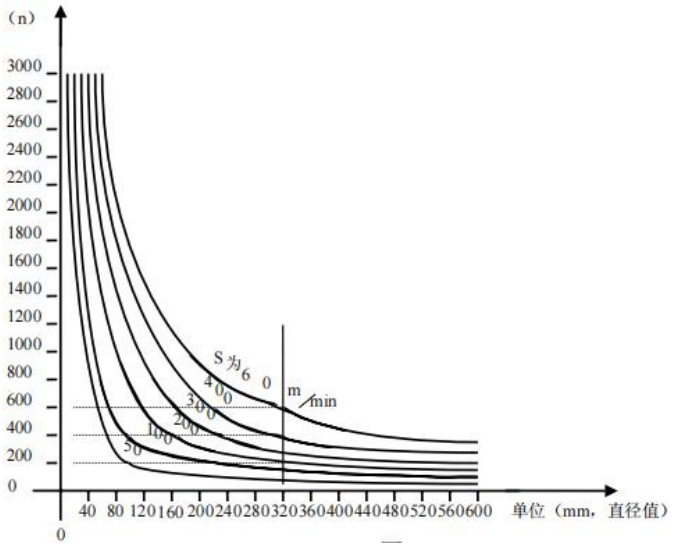


Figure 3-45

Constant linear speed control, only in the cutting feed (interpolation) process with the programming track of the X axis absolute coordinate value of the absolute value of the change of the spindle speed, for G00 fast movement, because not the actual cutting, the spindle speed at this time according to the G00 program segment end position to calculate the spindle speed and output, the G00 execution process of the spindle speed will remain unchanged.

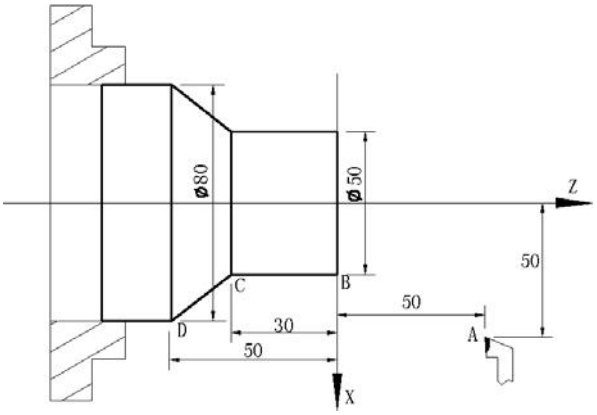
Constant line speed control, the workpiece coordinate system Z coordinate axis must be with the spindle axis (workpiece rotation axis) coincide, otherwise, the actual line speed will be with the given line speed is not consistent.

When the constant linear speed control is effective, G50 S xxxx can limit the maximum spindle speed (rpm). when the spindle speed calculated according to the linear speed and x-axis coordinate value is higher than the setting value of G50 S xxxx, the actual spindle speed is the maximum spindle speed limit value (I .e. the setting value of G50 S xxxx). When the system is powered on, the maximum spindle speed limit value is not set, and the maximum spindle speed limit function is invalid. The maximum speed limit defined by G50 S xxxx is maintained before re-designation. The maximum speed limit function is valid in G96 state. In G97 state, the maximum spindle speed set by G50 xxxx is not limited, but the spindle still has the maximum speed limit (I .e. the maximum output is 10V analog voltage).

Special attention is required: If the execution **G50 S0**, The spindle speed will be limited **99** turn / points (by transmission parameters **no. 116** setting).

The 116 number of the transmission parameter is the lower limit of the spindle speed when the constant line speed is controlled. When the spindle speed calculated by the line speed and the X-axis coordinate value is lower than this The value is that the actual spindle speed is limited to the lower spindle speed limit.

example:



Procedure: O0014;

M3 G96 S300; (spindle forward rotation, constant linear speed control is effective, linear speed is 300 m/min) G0

X100 Z100; (Move quickly to point a, spindle speed is 955 r/min during movement) G0

X50 Z0; (Move quickly to point B, spindle speed is 1910 r/min during movement) G1

W-30 F200; (cutting from point B to point c, the spindle speed during cutting is constant at 1910 r/min)

X80 W-20 F150; (from point C to point D, the spindle speed changes linearly from 1910 r/min to 1194 r/min)

G0 X100 Z100; (Quickly return to point A, the spindle speed is 955 r/min during movement)

M30;

Note 1: In the G96 state, the commanded S value is maintained even in the G97 state. When returning to the G96 state, its value is restored, for example:

G96 S50; (cutting line speed 50 m/min) G97

S1000; (spindle speed 1000r/min) G96 G01

X200; (cutting line speed 50 m/min)

Note 2: When the machine is locked (the X and Z axes do not move when the X and Z axis motion commands are executed), the constant linear speed control function is still effective;

Note 3: During thread cutting, the constant line speed control function is invalid, but in order to ensure the thread processing accuracy, the constant line speed control should not be used during thread cutting, and the thread cutting should be carried out in G97 state;

Note 4: When changing from G96 state to G97 state, if there is no instruction S (rev/min) in

G97 program segment, then the final rotation speed of G96 state is used as the S instruction of G97 state, that is, the rotation speed of the spindle will not change at this time;

Note 5: In constant linear speed control, when the spindle speed calculated from the cutting linear speed is higher than the maximum speed of the current spindle gear (system transmission parameters **107-111 No** .), the spindle speed at this time is limited to the maximum speed of the current spindle gear.

Feed per minute G98, feed per turn G99

Command format: G98 F xxxx ;(F0001 ~ F8000, leading zero can be omitted, given feed speed per minute, mm/minute)

Command function: in mm/sub-unit given cutting feed speed, G98 as modal G command. If it is currently G98 mode, G98 may not be entered.

Command format: G99 F xxxx ;(F0.0001 ~ F500, leading zero can be omitted)

Instruction function: in mm/conversion unit given cutting feed speed, G99 G99 as modal G instruction. If it is currently G99 mode, you can not enter G99. When the system executes G99 F xxxx, the F command value (mm /rev) is multiplied by the current spindle speed (rev/min) Product as the command feed speed to control the actual cutting feed speed, the spindle speed changes, the actual cutting feed speed. with the change. Using G99 F xxxx to give the cutting feed per revolution of the spindle, uniform cutting patterns can be formed on the surface of the workpiece road. To perform machining in the G99 mode, the machine must be fitted with a spindle encoder.

G98 and G99 are the same group of modal G instructions, and only one can be valid at the same time. G98 is the initial state G command, which defaults when the system is powered on.
G98 is valid.

Conversion formula for feed per revolution and feed per minute:

$$F_m = F_r \times S$$

Of which: F m: Feed per minute (mm/min);

F r: Feed per revolution (mm/r);

s: Spindle speed (r/min).

After executing the F instruction, the value of F remains unchanged. After execution of F0, the feed speed is 0. During system reset and emergency stop, the F value remains unchanged. Feed times rate power-down memory.

Relevant parameters:

System transmission parameter 91 No: cutting feed rate upper limit (X axis, Z axis is the same, for the X axis is diameter change/min)

System transmission parameter No. 97: exponential acceleration and deceleration time constant for cutting feed and manual feed;

Note: In the G99 mode, when the spindle speed is lower than 1 r/min, the cutting feed speed will be uneven; when the spindle speed fluctuates, the actual cutting feed speed there will be a follow-up error. In order to ensure the processing quality, it is recommended that the spindle speed selected during processing should not be lower than the minimum speed of the spindle servo or inverter output effective torque.

macro instruction

Provides macro instructions similar to high-level languages, user macro instructions can realize variable assignment, arithmetic operations, logical judgment and condition transfer, which is conducive to the preparation of special parts of the processing program, reduce manual programming for tedious numerical calculations, streamlining the user program.

macro variable

(**1**) Use of variables

A variable can instruct an address value in a program. Variable values can be assigned by program instructions or set directly with the keyboard. Multiple variables can be used in a program, and these variables are distinguished by variable numbers.

- Representation of variables

  Use "#" + variable number to indicate;

  Format:# I (I = 200,202,203,...);

  example:#205,#209,#225.

- References to variables

1. Replace the address with a variable.

   Format: <Address> + "# I" or <Address> + "-# I", which means that the value of the variable or the negative value of the variable value is used as the address value. Example: F#203… when #203=15, same function as F15 instruction;
   Z-#210… When #210=250, the function is the same as that of the Z-250 command;
   G#230… When #230=3, the function of the G3 instruction is the same.

2. Replace the variable number with a variable.

Format: "#" + "9" + replacement variable number.

Example: When #200=205, when #205=500,

X#9200 and X500 instructions function the same;

X-#9200 and the X-500 command function the same.

Note **1**: Addresses **O** and **N** cannot refer to variables. Cannot be programmed with **O#200**, **N#220**;

Note **2**: If the maximum instruction value specified by the address is exceeded, it cannot be used. For example, when **#230=120**, **M#230** exceeds the maximum instruction value.

(**2**) Types of Variables

Depending on the variable number, variables are divided into common variables and system variables, which have different uses and properties.

- Common Variables

  Common variables are #200 ~#231. Common variables are common in programs. That is, the variables and operation results defined in the program 1 are applied to the program 2 and the program 3.
  Common variables are #200 ~#231, and the values of all variables are maintained after power failure.

- System Variables

  The purpose of the system variable is fixed in the system. The input signals of the system variable interface are #1000 ~#1015, and the output signals of the interface are

#1100 ~#1105;

The system variable interface input/ output signal is determined by the PLC, the factory standard ladder diagram of the system variable input signal #1000 ~#1015, output signal #1100 ~#1105 PLC signal address, see the seventh chapter standard ladder diagram function configuration.

Operation Command and Transfer Command G65

General instruction format:

G65 H m P# I Q# j R# k;

Where: m: represents an operation command or a transfer command function.

I: The name of the variable in which the operation result is stored.

j Variable name of the operation 1, can also be a constant.

k Name: The variable name of the operation 2, or it can be a constant.

Instruction meaning:# I = #j O # k

operation symbol, determined by Hm

Example: P#200 Q#201 R#202… #200 = #201 O #202;
P#200 Q#201 R15… #200 = #201 O 15;
P#200 Q-100 R#202…..#200 = -100 O #202;

Note: ● If the variable value does not include decimal point, the unit is 0.001mm; Example:#100=30, X#100 = X0.03 mm;

● Variables are directly expressed as constants without "#";

macro instruction table

Command Format	Function	Definition
G65 H01 P# I Q#j	Assignment	# I = # j; Assign the value of variable j to variable I
G65 H02 P# I Q #j R #k;	decimal addition operation	# I = # j + # k
G65 H03 P# I Q #j R #k;	decimal subtraction operation	# I = # j - # k
G65 H04 P# I Q #j R #k;	decimal multiplication	# I = # j×# k
G65 H05 P# I Q #j R #k;	decimal division operation	# I = # j÷# k
G65 H11 P# I Q #j R #k;	Binary addition (or operation)	# I = # j OR # k
G65 H12 P# I Q #j R #k;	Binary Multiplication (AND)	# I = # j AND # k
G65 H13 P# I Q #j R #k;	Binary XOR	# I = # j XOR # k
G65 H21 P# I Q #j;	Decimal square	# I = # j
G65 H22 P# I Q #j;	decimal take absolute value	# I = |# j |
G65 H23 P# I Q #j R #k;	decimal remainder	# I = remainder of (#j÷# k)
G65 H24 P# I Q #j;	Decimal to binary	# I = BIN(# j)
G65 H25 P# I Q #j;	Binary to Decimal	# I = DEC(# j)
G65 H26 P# I Q #j R #k;	decimal multiplication and division	# I = # i×# j÷# k
G65 H27 P# I Q #j R #k;	compound square root	# I = $\#j^2$ $\#k^2$
G65 H31 P# I Q #j R #k;	Sine	# I = # j×sin(# k)
G65 H32 P# I Q #j R #k;	Cosine	# I = # j×cos(# k)
G65 H33 P# I Q #j R #k;	Tangent	# I = # j×tan(# k)
G65 H34 P# I Q #j R #k;	Anyway cut	# I = ATAN(# j / # k)
G65 H80 P n;	Unconditional Transfer	Jump to segment n
G65 H81 P n Q #j R #k;	Conditional Transfer 1	If# j = # k jump to segment n, otherwise sequential execution
G65 H82 P n Q #j R #k;	Conditional Transfer 2	If# j≠# k jump to segment n, otherwise sequential execution

Command Format	Funct ion	Defin ition
G65 H83 P n Q #j R #k;	Conditional Transfer 3	If# j>=# k jump to segment n, otherwise sequential execution
G65 H84 P n Q #j R #k;	Conditional Transfer 4	If# j <# k jump to segment n, otherwise sequential execution
G65 H85 P n Q #j R #k;	Conditional Transfer 5	If# j ≥# k jump to segment n, otherwise sequential execution
G65 H86 P n Q #j R #k;	Conditional Transfer 6	If# j ≤# k jump to segment n, otherwise sequential execution
G65 H99 P n;	Generating user alarm	Generate (500 + n) user alarm

1 Operation command

Variable assignment:# I = # J

G65 H01 P# I Q#J

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

G65 H01 P #201 Q#210; (#201 = #210)

G65 H01 P #201 Q-#202; (#201 = -#202)

2) Decimal addition operation:# I = # J +# K

G65 H02 P# I Q#J R#K

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

3) Decimal subtraction:# I = # J-# K

G65 H03 P# I Q#J R# K

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

4) Decimal multiplication:# I = # J ×# K

G65 H04 P# I Q#J R#K

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

5) Decimal division:# I = # J ÷# K

G65 H05 P# I Q#J R#K

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

6) binary logic plus (or):# I = # J. OR.# K

G65 H11 P# I Q#J R#K

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

7) binary logic multiplication (and):# I = # J AND.# K

G65 H12 P# I Q#J R#K

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

8) Binary XOR:# I = # J. X OR.# K

G65 H13 P# I Q#J R#K

(example) G65 H13 P#201 Q#202 R#203; (#201 = #202 .X OR. #203)

9) Decimal square:# $\sqrt{\#J}$

I =

G65 H21 P# I Q#J

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

10) Decimal takes absolute value:# I = | # J |

G65 H22 P# I Q#J

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

11) decimal remainder:# I = # J-TRUNC(#J/#K)×# K,TRUNC: decimal part

G65 H23 P# I Q#J R#K

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

12) Decimal to binary:# I = BIN (# J)

G65 H24 P# I Q#J

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

13) Binary to decimal:# I = BCD (# J)

G65 H25 P# I Q#J

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

14) decimal multiplication and division:# I =(# I×# J)÷# K

G65 H26 P# I Q#J R# k

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

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

G65 H27 P# I Q#J R#K

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

16) Sine:# I = # J • SIN(# K)(Unit: ‰ degree)

G65 H31 P# I Q#J R#K

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

17) cosine:# I = # J • COS(# K)(unit: ‰ degree)

G65 H32 P# I Q#J R# k

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

18) Tangent:# I = # J • TAM(# K)(unit: ‰ degree)

G65 H33 P# I Q#J R# K

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

19) arctangent:# I = ATAN(# J /# K)(unit: ‰ degree)

G65 H34 P# I Q#J R# k

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

]

Note **1**: The unit of (**P**)~(**S**) is designated for degree, and the unit is **1** ‰ degree;

Note **2**: In each operation, the variable value is only an integer, and the decimal point of the operation result is rounded off, and the variable value unit is **μm**;

Note **3**: The variable value is in the range of -2^{32}~ $+2^{32}$ -1, but only -9999999~9999999 can be displayed correctly. When it exceeds the above range, it will be displayed.

* * * * * * *.

2 Transfer orders

1) Unconditional transfer

G65 H80 Pn; **n**: sequence number

(Example) G65 H80 P120;(Go to N120 program section)

2) conditional transfer #1 j. EQ.# K (=)

G65 H81 Pn Q#J R# K; **n**: sequence number

(Example) G65 H81 P1000 Q#201 R#202;

When #201 = #202, go to the N1000 program segment, when #201 #202, the sequence is executed.

3) conditional transfer #2 j. NE.# K (≠)

G65 H82 Pn Q#J R# K; **n**: sequence number

(Example) G65 H82 P1000 Q#201 R#202;

When #201 #202, go to the N1000 program segment, when #201 = #202, the program sequence execution.

4) conditional transfer #3 j. GT.# K (>)

G65 H83 Pn Q#J R# K; **n**: sequence number

(Example) G65 H83 P1000 Q#201 R#202;

When #201 > #202, go to the N1000 program segment, when #201 #202, the program sequence execution.

5) conditional transfer #4 j. LT.# K (< =)

G65 H84 Pn Q#J R# K; **n**: sequence number

(Example) G65 H84 P1000 Q#201 R#202;

When #201 #202, go to the N1000 program segment, when #201 #202, the program sequence execution.

6) conditional transfer #5 j. GE.# K (≥)

G65 H85 Pn Q#J R# K; **n**: sequence number

(Example) G65 H85 P1000 Q#201 R#202;

When #201 #202, go to the N1000 program segment, when #201 #202, the order of execution.

7) conditional transfer #6 j. LE.# K (≤)

G65 H86 Pn Q#J R# K; **n**: sequence number

(Example) G65 H86 P1000 Q#201 R#202;

When #201 #202, go to the N1000 program segment, when #201 > #202, sequential execution.

8) P/S alarm occurs

G65 H99 Pi; **i**: alarm number **+500**

(Example) G65 H99 P15;

P/S alarm 515 occurs.

Note to note **1:** Sequence numbers can be specified by variables. For example, **G65 H81 P#200 Q#201 R#202**; When the condition is met, the program moves to the program segment of the sequence number specified by **#200**.

Chamfer function

The chamfering function is to insert a straight line or arc between the two wheel profiles, so that the tool can smoothly transition from one contour to another 1. With a straight line and arc two kinds of chamfer function.

Straight line chamfer

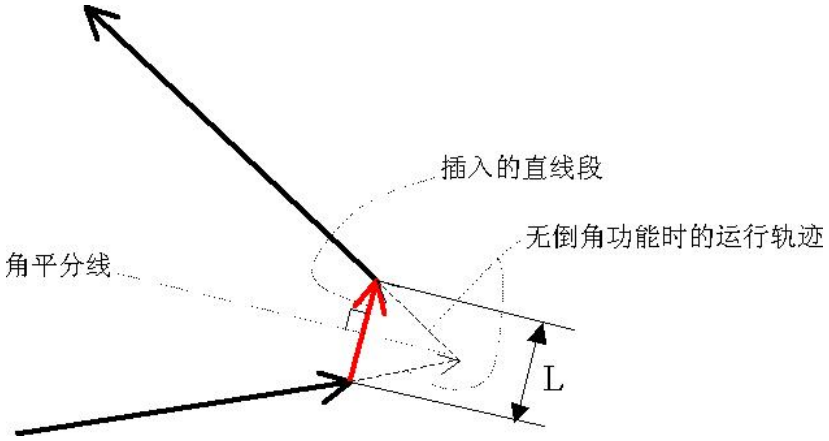
Straight line chamfer: a straight line is inserted between straight line contours, between arc contours, and between straight line contours and arc contours. The command address of the straight line chamfering is L, and the data after the command address is the length of the chamfered straight line. Straight chamfers must be used in the G01, G02, or the G03 command segment.

A. Straight line to straight line

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

G01 X(U)_ Z(W)_ ;

Command function: insert a straight line segment into two straight line interpolation command segments.



B. straight line connected to arc

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

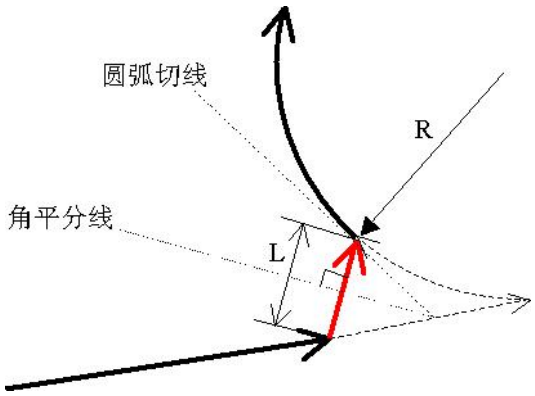
G02/G03 X(U)_ Z(W)_ R _;

Or

G01 X(U)_ Z(W)_ L_;

G02/G03 X(U)_ Z(W)_ I_ K_;

command function: insert a straight line segment between straight line and circular interpolation command.



C. arc to arc

Command format: G02/G03 X(U)_ Z(W)_ R_ _L_;

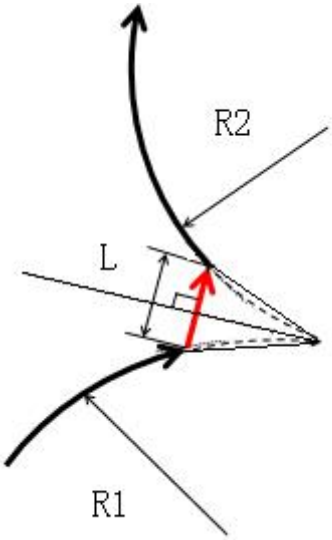
G02/G03 X(U)_ Z(W)_ I_ K _;

or

G02/G03 X(U)_ Z(W)_ I_ K_ L_;

G02/G03 X(U)_ Z(W)_ R_;

Command function: insert a straight line segment between two circular interpolation commands.



D. Arc connected to straight line

Command format: G02/G03 X(U)_ Z(W)_ R_ _L_;

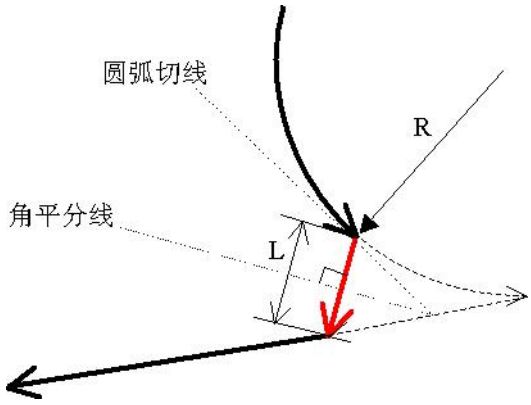
G01 X(U)_ Z(W) _;

Or

G02/G03 X(U)_ Z(W)_ I_ K_ L_;

G01 X(U)_ Z(W)_;

Command function: insert a straight line segment between arc and straight line interpolation command.



arc chamfer

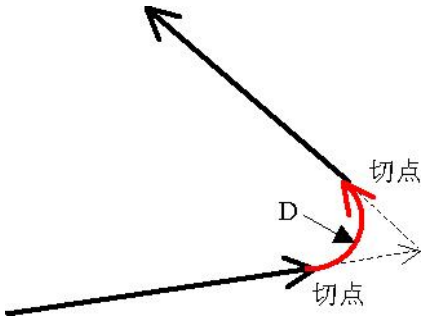
Arc chamfer: insert 1 arcs between straight line contours, between arc contours, between straight line contours and arc contours, and make tangent transition between arc and contour. The command address of the arc chamfering is D, and the data after the command address D is the radius of the chamfered arc. The arc chamfer must be in G01, G02 or
Used in the G03 instruction segment.

E. Straight line to straight line

Instruction format: G01 X(U) Z(W)_ D;

G01 X(U)_ Z(W) _;

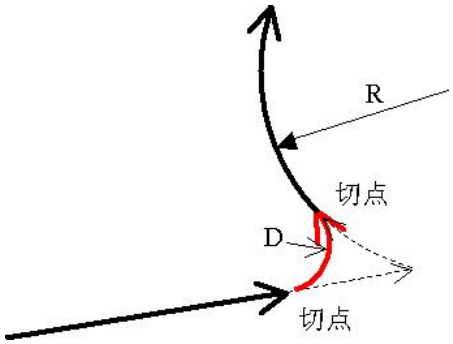
Instruction function: insert a circular arc into the two linear interpolation segments. The inserted circular arc segments are tangent to the two straight lines, and the radius is the data after the instruction address D.



F. straight line connected to arc

Command format: G01 X(U)_ Z(W)_ D_ ;
G02/G03 X(U)_ Z(W)_ R_;
or
G01 X(U)_ Z(W)_ D_;
G02/G03 X(U)_ Z(W)_ I_ K_;

Command function: insert an arc at the junction of a straight line and an arc, the inserted arc segment is tangent to both the straight line and the arc, and the radius is the command address data after D.



G. arc to arc

Command format: G02/G03 X(U)_ Z(W)_ R_ D_;G02/G03 X(U)_ Z(W)_ R_.

or

G02/G03 X(U)_ Z(W)_ R_ D_;G02/G03 X(U)_ Z(W)_ I_ K_.

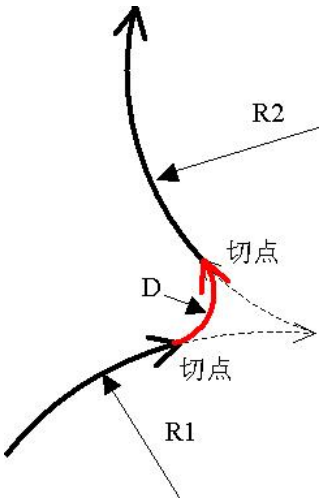
or

G02/G03 X(U)_ Z(W)_ I_ K_ D_;G02/G03 X(U)_ Z(W)_ I_ K_.

or

G02/G03 X(U)_ Z(W)_ I_ K_ D_;G02/G03 X(U)_ Z(W)_ R_.

command function: insert a circular arc between two circular arc interpolation segments. the inserted circular arc segment is tangent to both circular arcs, and the radius is the data after command address d.



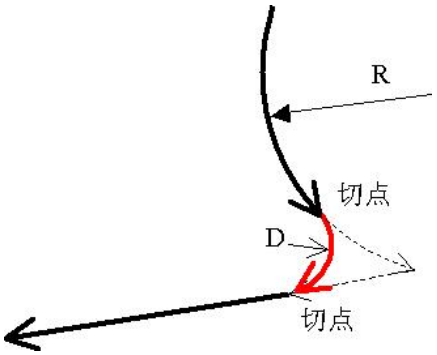
D. arc connected to straight line

Command format: G02/G03 X(U)_ Z(W)_ R_ D_;G01 X(U)_ Z(W)_.

or

G02/G03 X(U)_ Z(W)_ I_ K_ D_;G01 X(U)_ Z(W)_.

Command function: insert a circular arc at the junction of circular arc and straight line. the inserted circular arc segment is tangent to circular arc and straight line, and the radius is the data after command address d.

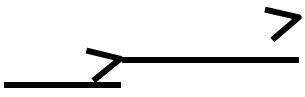


Special circumstances

In the following cases, the chamfer function is invalid or alarm.

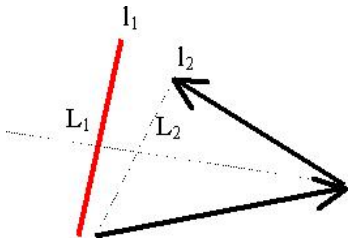
1) When chamfering straight line

H. When two interpolated straight line segments are on the same straight line, the chamfering function is invalid.



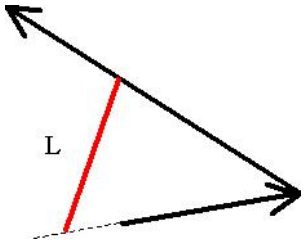
I. If the chamfer line length is too long, the CNC generates an alarm.

As shown in the figure below, l_1 is a chamfered straight line with a length of L_1;l_2 the third side of the triangle formed by connecting two interpolated straight lines, with a length of L_2, when L_1 greater than L_2 CNC generates alarm.



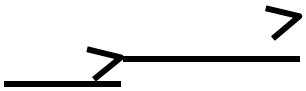
J. A section of straight line (arc) is too short, alarm

As shown in the following figure, the length of the chamfering straight line is L. After calculation, the other end of the chamfering straight line is not on the interpolation straight line (on the extension line of the interpolation straight line), and CNC generates an alarm.



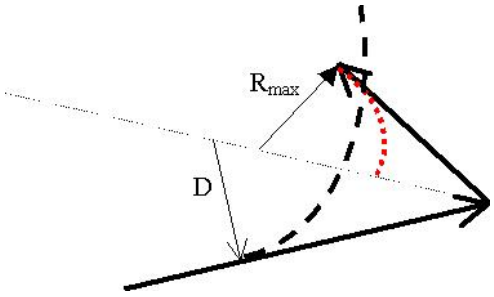
2) During arc chamfering

A. When two interpolation line segments are on the same line, the arc chamfering function is invalid.

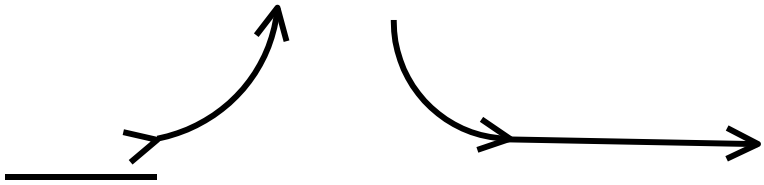


B. If the radius of the chamfered arc is too large, the CNC will generate an alarm.

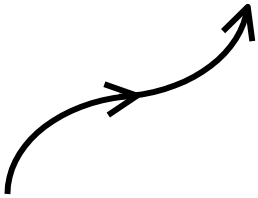
As shown in the following figure, the radius of the chamfered arc is D, the maximum arc radius of the tangent of the two straight lines is Rmax,Rmax is less than D, and CNC generates an alarm.



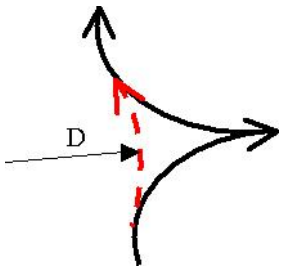
C. When a straight line is tangent to an arc or an arc is tangent to a straight line, the arc chamfering function is invalid.



D. When the arc is tangent to the arc, the arc chamfering function is invalid;



However, if the arc is tangent like the following figure, the arc chamfering function is effective.



Tip radius compensation (G41, G42)

Application of nose radius compensation

Overview

The part processing program is generally based on a certain point of the tool (usually an imaginary tool tip, as shown in point A of Figure 4-1) according to the part drawing. But the actual processing of the turning tool, due to the process or other requirements, the tip is often not 1 imaginary point, but a circular arc. When cutting, there is a deviation between the actual cutting point and the ideal cutting point, which will cause over-cutting or less cutting and affect the accuracy of the part. Therefore, the nose radius compensation is carried out in the processing to improve the precision of the parts.

Fig. 4-1 Cutter

The part shape of the trajectory offset a tool tip radius of the method is the B- type tool compensation method, this method is simple, but in the implementation of the 1 program segment is completed, only to deal with the next 1 program segment of the motion trajectory, so in the two programs of the intersection will produce over-cut and other phenomena.

In order to solve the above problems and eliminate the error, it is necessary to establish the C type tool compensation method. When the C- type tool compensation method is read into the 1 program segment, it is not immediately executed, but the next 1 program segment is read into, and the corresponding motion trajectory (transfer vector) is calculated according to the connection of the intersection of the two program segments. Since two program segments are read for preprocessing, the C -type tool compensation method can compensate more accurately on the contour. 4-2.

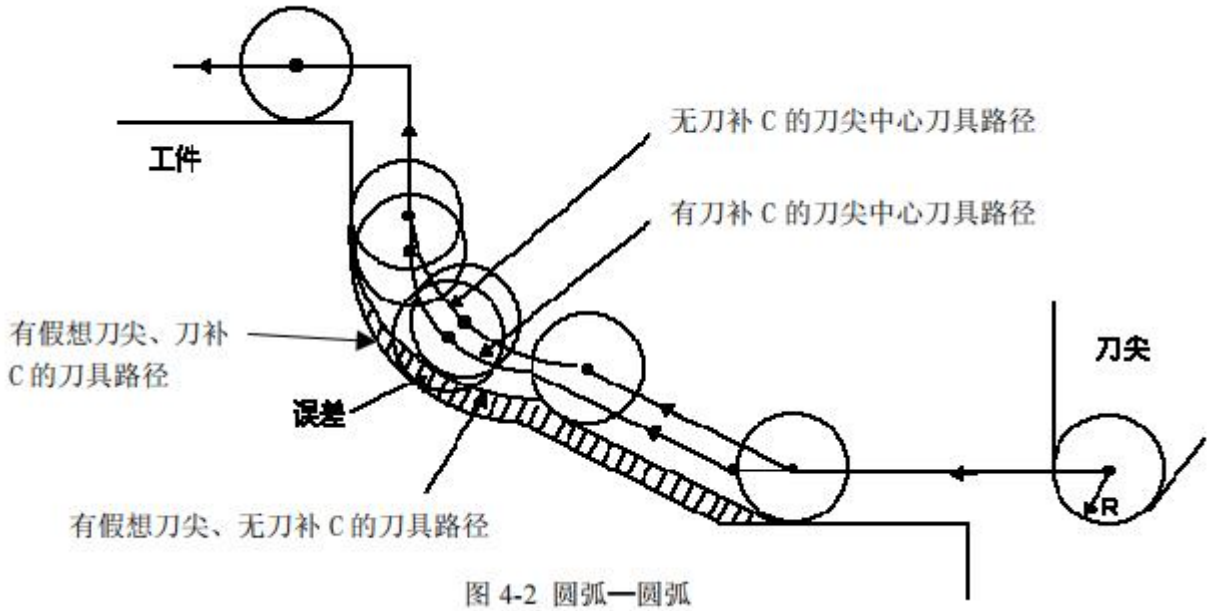
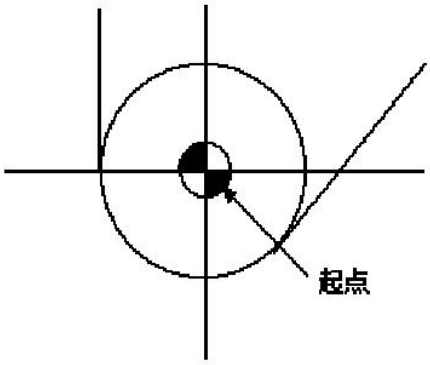


图 4-2 圆弧—圆弧

imaginary nose direction

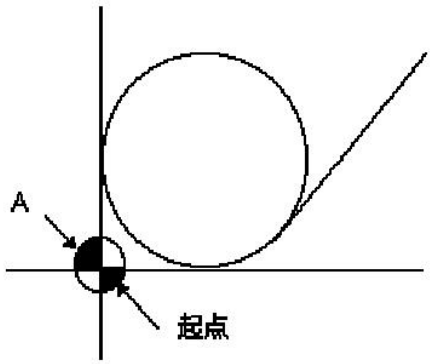
The setting of the imaginary tip is because it is generally difficult to set the center of the tip radius at the starting position, as shown in Figure 4-3. However, it is relatively easy to set the imaginary tip at the starting position, as shown in Figure 4-4. The tip radius may not be considered during programming. 4-5, 4-6 are respectively with tip center programming and imaginary tip programming, using tip radius compensation and not using tip radius compensation tool path diagram comparison.



用刀尖中心编程时

图 4-3

If you don't use tip radius compensation, The tip center trajectory will be the same as the programmed trajectory



用假想刀尖编程时

图 4-4

if you use tip radius compensation, Will achieve precision cutting

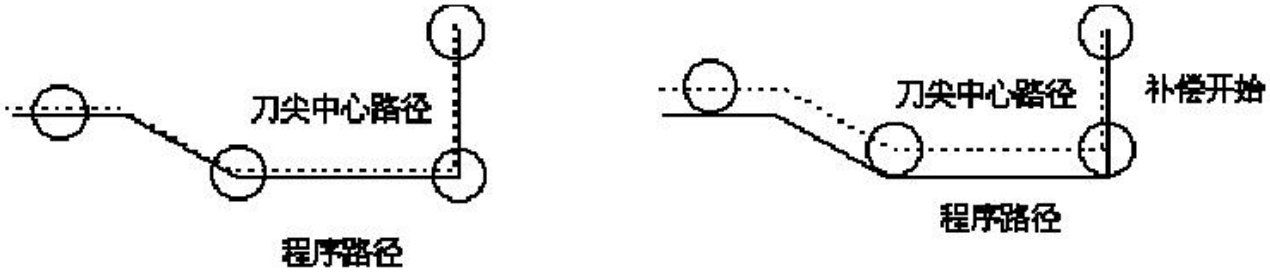


Figure 4-5 Tool path when programming with tool tip center

Without nose radius compensation, hypothetically The nose trajectory will be the same as the programmed trajectory

using nose radius compensation, will achieve precision cutting

Figure 4-6 Tool path when programming with imaginary tool tip

In the programming process, the tool is imagined as a point, and the actual cutting edge due to process requirements or other reasons can not be an ideal point. This kind of machining error caused by the cutting edge is not 1 ideal point but a circular arc can be eliminated by the tool tip arc radius compensation function. In actual processing, the imaginary tip point and the center point of the tip arc have different positional relationships, so the tip direction of the imaginary tip should be correctly established (that is, which position of the tool is the tool).

Looking from the tip center to the direction of the imaginary tip, the imaginary tip number is determined by the direction of the tool in the cutting. A total of 10(T0 ~ T9) settings of the imaginary tip, a total of 9 directions to express the positional relationship. Special attention should be paid to the fact that even the same tool tip direction number indicates different tool tip directions in different coordinate systems (rear tool seat coordinate system and front tool seat coordinate system), as shown in the following figure. The figure illustrates the relationship between the tip and the starting point, and the end point of the arrow is the imaginary tip. The situation of the rear tool seat coordinate system T1 ~ T8 is shown in Figure 4-7. The situation of the front tool seat coordinate system T1 ~ T8 is shown in Figure 4-8. T0 and T9 are the case when the tip center coincides with the starting point, as shown in Figure 4-9.

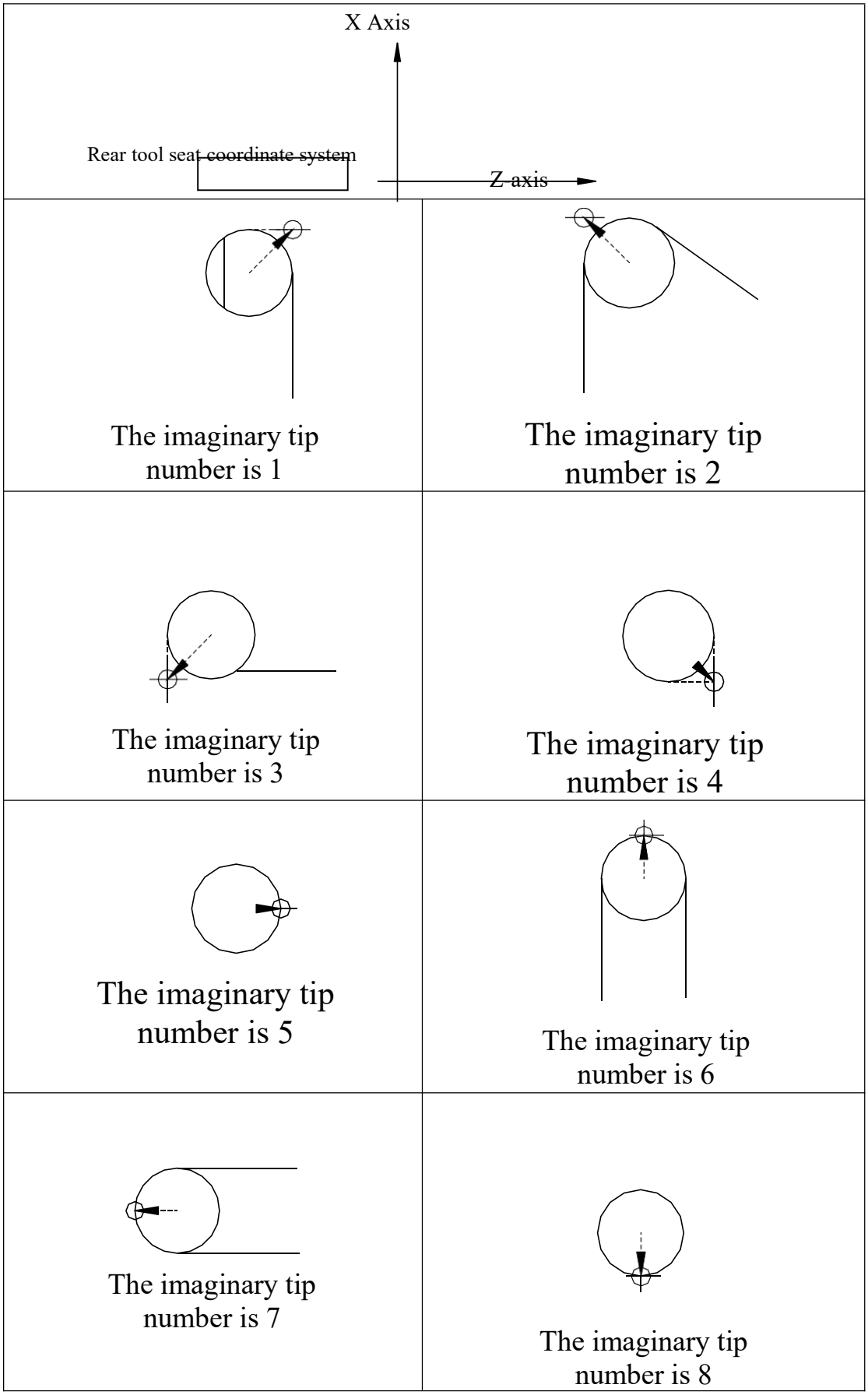


Fig. 4-7 Imaginary tool tip number in rear tool holder coordinate system

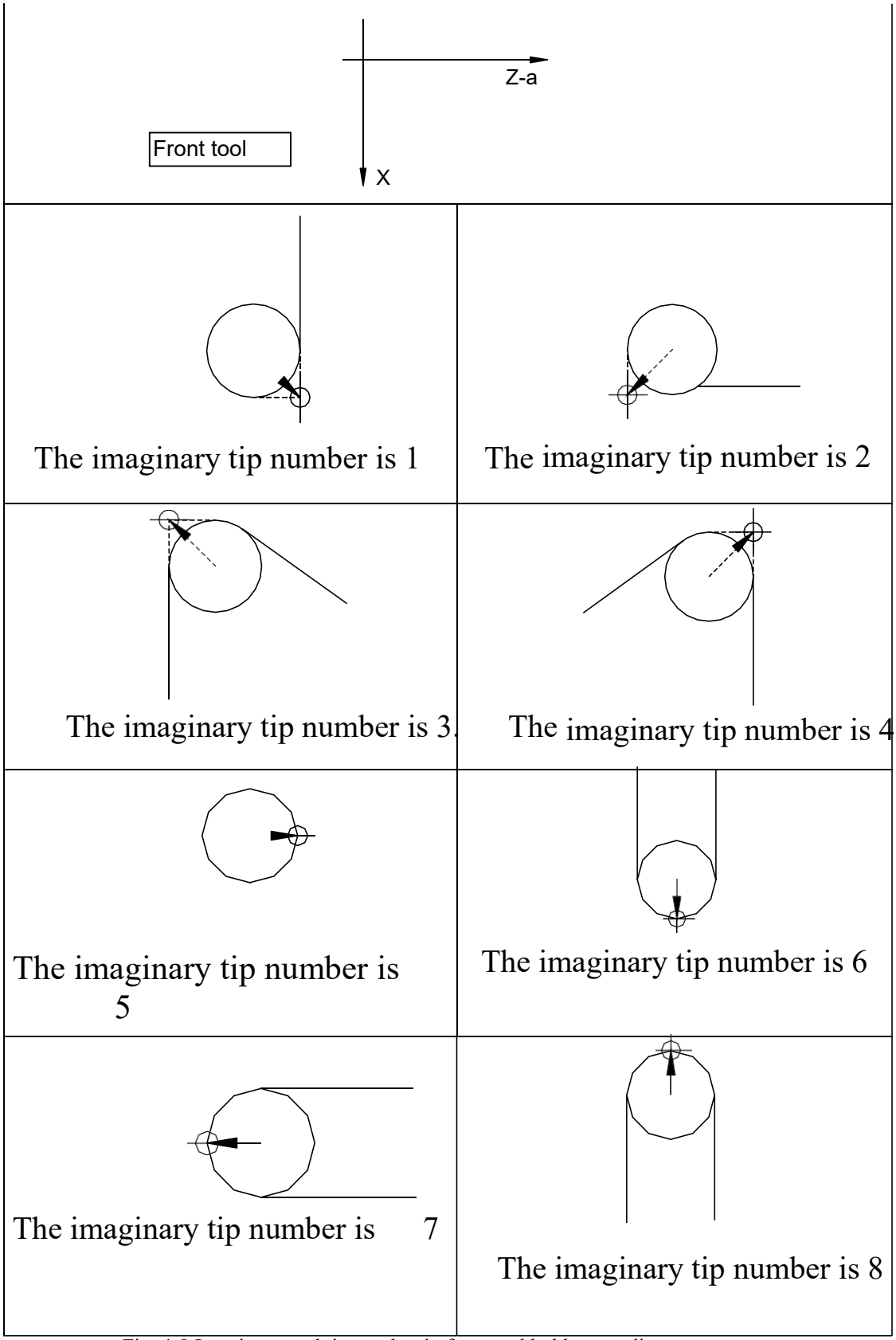


Fig. 4-8 Imaginary tool tip number in front tool holder coordinate system

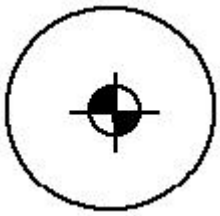
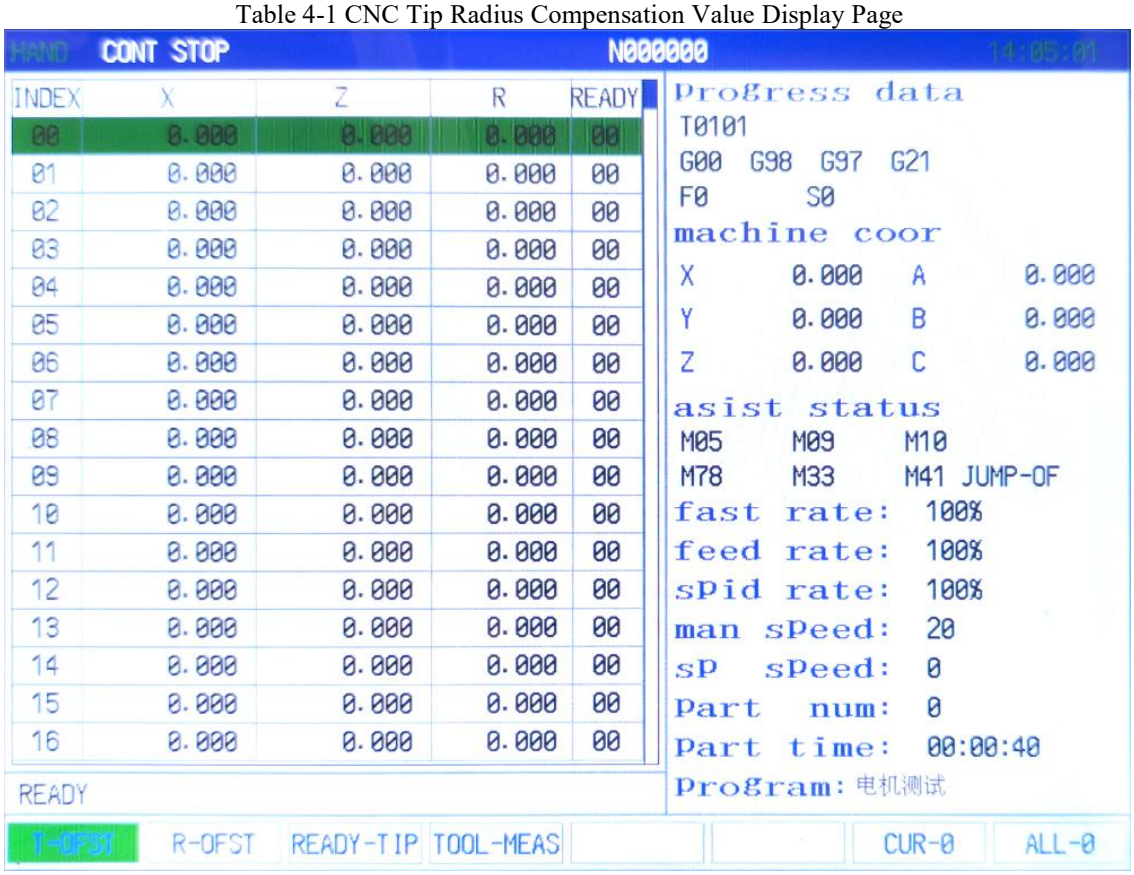


Fig. 4-9 The center of the tip is consistent with the starting point

Setting of compensation value

The imaginary tip number and tip radius value of each knife must be pre-set before applying the C- knife patch. The nose radius compensation value is set under the offset page (see Table 4-1), R is the nose radius compensation value, and T is the imaginary nose number.

Table 4-1 CNC Tip Radius Compensation Value Display Page



Note: The tool offset value in X direction can be specified by diameter or radius value, which is set by ORC of bit4 bit of parameter No.004. When ORC = 1, the offset value is expressed by radius, and when ORC = 0, the offset value is expressed by diameter.

In the knife operation should pay special attention to, when the choice of Tn(n = 0 ~ 9) imaginary tip, the knife point must also be Tn(n = 0 ~ 9) imaginary knife point. As shown in fig. 4-10, different tool setting methods are used when T0 and T3 tool tip points are selected in the rear tool holder coordinate system. taking the tool holder center as the standard point, the offset value from the standard point to the tool tip radius center (when the imaginary tool tip is T0) and the offset value from the standard point to the imaginary tool tip (when the imaginary tool tip is T3) are different for the same tool. Measuring the distance from the standard point to the imaginary tip is much easier than measuring the distance from the standard point to the center of the tip radius. Therefore, the tool offset value is usually set by the distance from the standard point to the imaginary tip (I. e., T3 tip direction is usually selected).

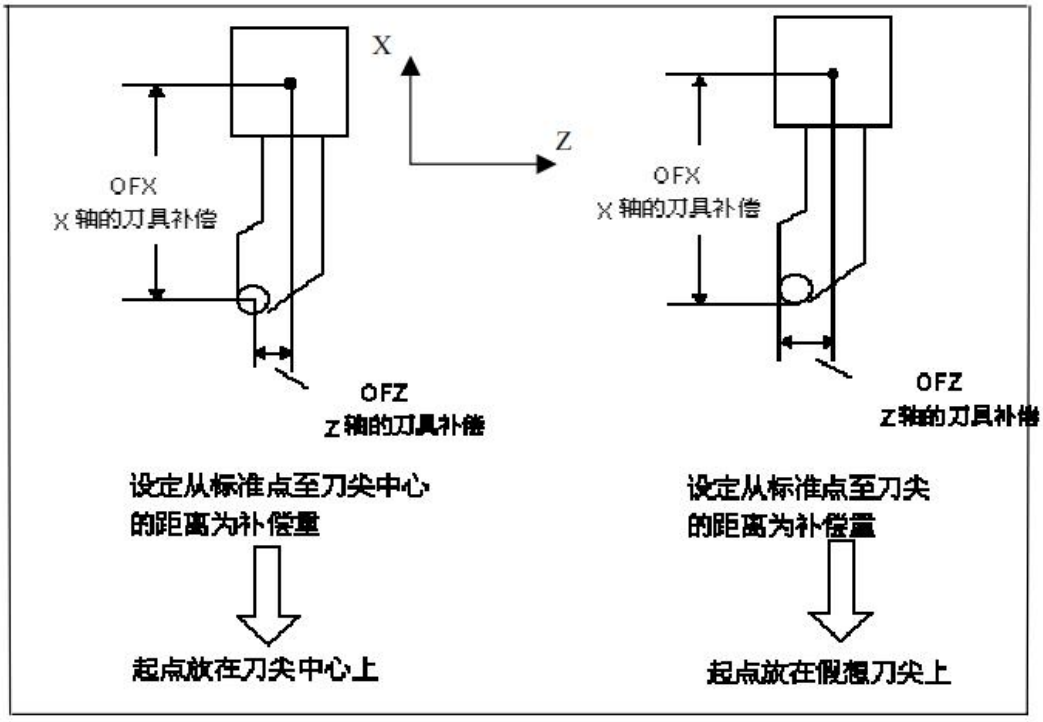


Fig. 4-10 Tool Offset Value with Tool Holter Center as Reference Point

Command Format

{ G40 / G41 / G42 } { G00 / G01 } X_ Z_ T ;

Instruction	Function Description	Remarks
G40	Cancel nose radius compensation	See Fig. 4-11 and Fig. 4-12 said. Ming
G41	G41 in the rear tool holder coordinate system is designated as left tool compensation, and G41 in the front tool holder coordinate system is designated as right tool compensation.	
G42	G42 in the rear tool holder coordinate system is designated as right tool compensation, and G42 in the front tool holder coordinate system is designated as left tool compensation.	

Compensation direction

To apply tool tip radius compensation, the direction of compensation must be determined according to the relative position of the tool tip and the workpiece, as shown in Figures 4-11 and 4-12.

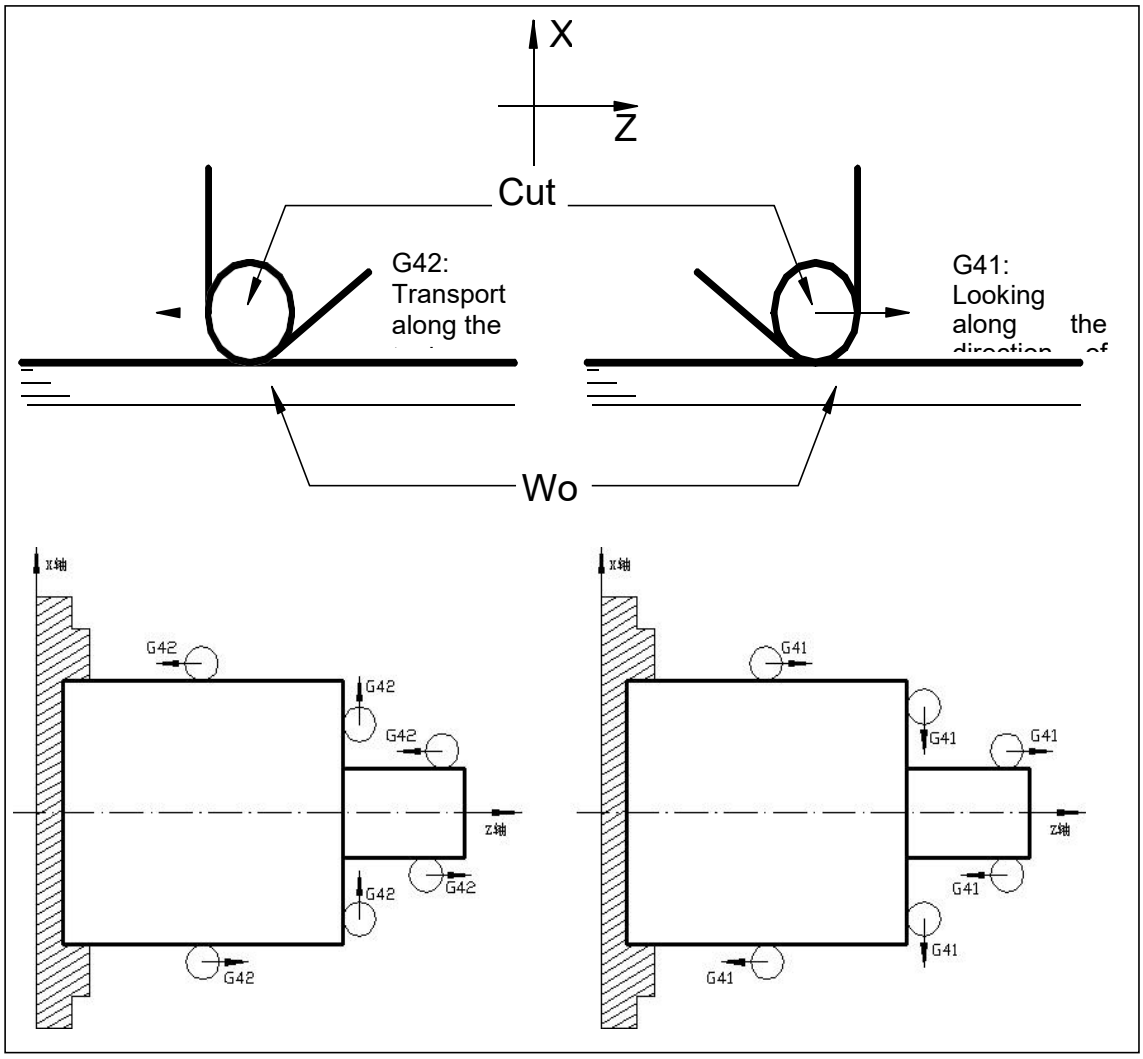


Fig. 4-11 Compensation Direction of Rear Tool Holter Coordinate System

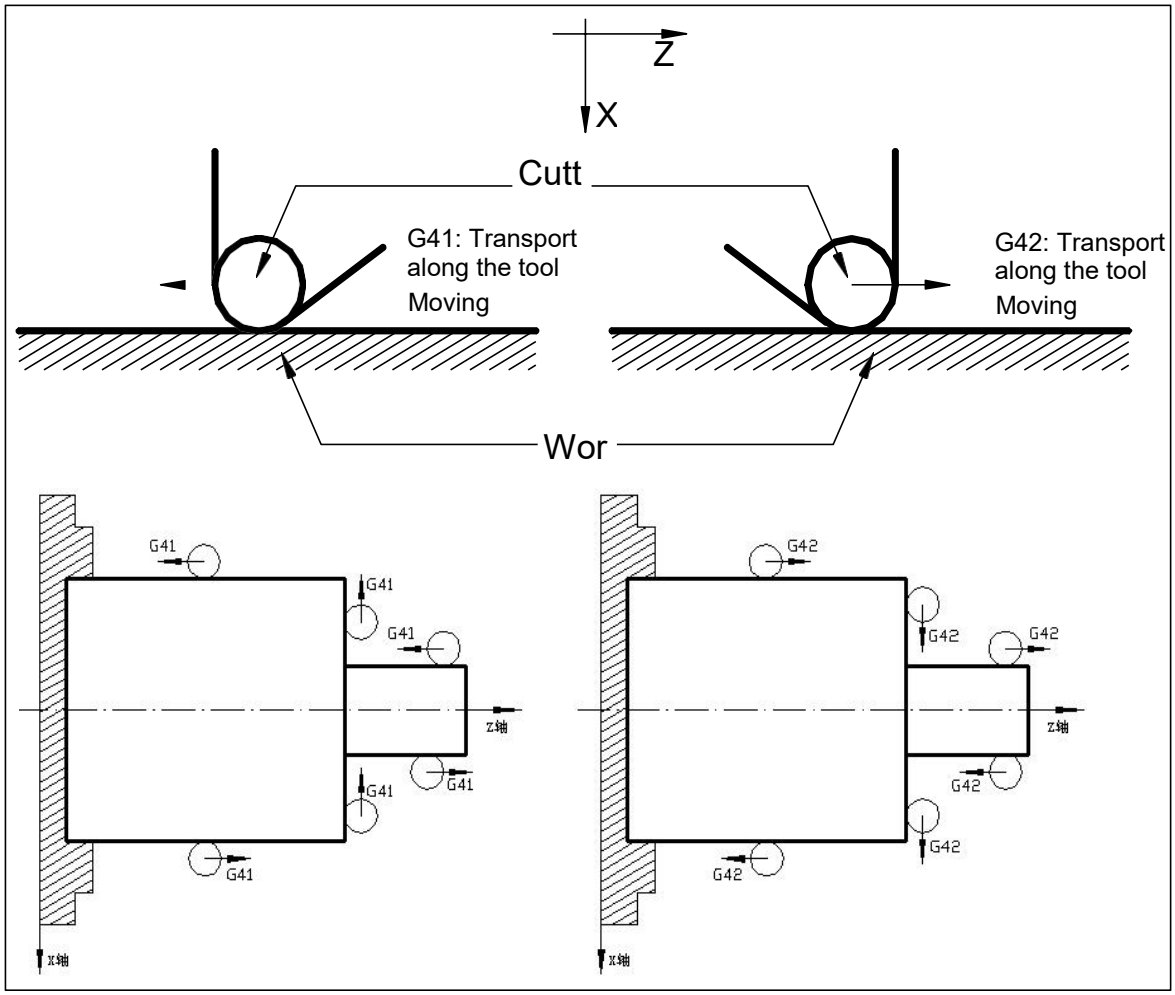


Fig. 4-12 Compensation direction of front tool holder coordinate system

Precautions

- In the initial state, the CNC is in the tool tip radius compensation cancellation mode. After executing the G41 or G42 command, the CNC starts to establish the tool tip radius compensation offset mode. At the beginning of the compensation, the CNC pre-reads the 2 program segments, and when the 1 program segment is executed, the next 1 program segment is stored in the tool radius compensation buffer memory. In single-segment operation, two program segments are read in, and the end of the first program segment is executed and then stopped. During continuous execution, two program segments are read in advance, so that the CNC is executing a program segment and two subsequent program segments.
- In the tip radius compensation, when two or more non-moving instructions of the program segment (such as auxiliary functions, pause, etc.), the tip center will move to the end of the previous program segment and perpendicular to the previous program segment program path position.
- In MDI mode, you cannot create or undo Cutter Supplement C.
- A negative value cannot be entered for the nose radius R value, otherwise the running track is wrong.
- The establishment and cancellation of tool tip radius compensation can only use G00 or G01 command, not arc command (G02 or G03). If specified, an alarm is generated.
- After pressing the RESET key or executing M30, the CNC will cancel the tool compensation C compensation mode.
- The G40 cancel offset mode must be specified before the end of the program. Otherwise, the tool path is offset by one nose radius value when executed again.
- Tool tip radius compensation is used in the main program and subroutine. Before calling the subroutine (I. e. before executing M98),CNC must establish tool compensation C again in the subroutine in compensation cancellation mode.
- G71, G72, G73, G74, G75, G76 commands do not perform nose radius compensation, temporarily cancel the compensation mode.
- G90 and G94 instructions are performing nose radius compensation. Both G41 and G42 are offset

by one nose radius (according to the imaginary nose number 0) for cutting.

Application example

Machining the part shown in Figure 4-13 in the front tool seat coordinate system. Use tool number T0202, nose radius R = 2, imaginary nose number T = 3.

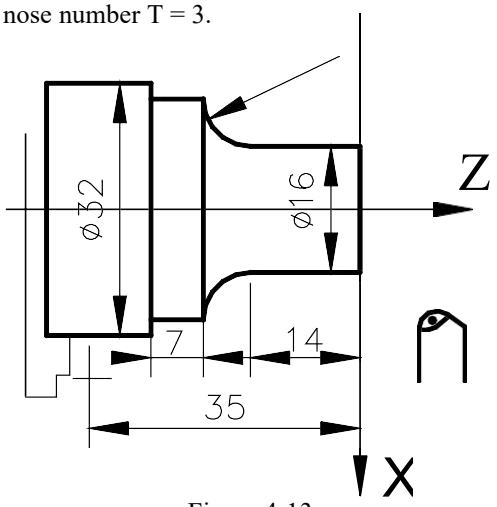


Figure 4-13

In the offset cancellation mode for the knife, after the knife is completed, usually the Z axis to offset a tool tip radius value, offset direction and imaginary tool tip direction and the knife point, otherwise in the knife will be over-cut a tool tip radius value.

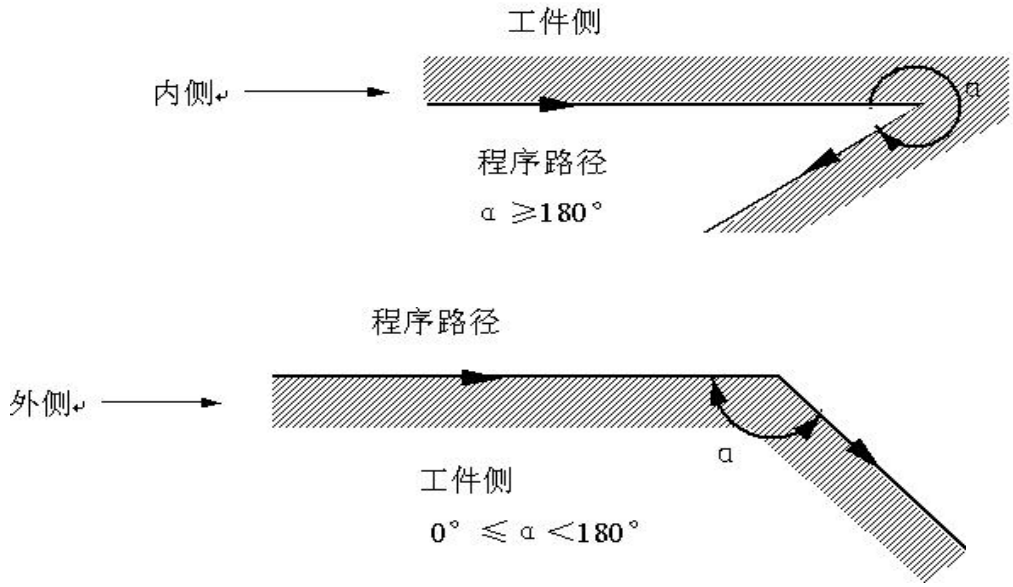
Under the knife compensation setting page, the radius of the knife tip R and the imaginary knife tip direction are set as follows:

```
Procedure:
G00 X100 Z50 M3 T0202 S600;          (Positioning, spindle opening, tool change,
                                      and tool compensation execution)
G42 G00 X0 Z3;                       (Establish tool tip radius compensation)
G01 Z0 F300;                         (Cutting start)
X16;
Z-14 F200;
G02 X28 W-6 R6
G01 W-7;
X32
Z-35;
G40 G00 X90 Z40;                     (Cancel tool tip radius compensation)
G00 X100 Z50 T0100
M30;
```

Tip Radius Compensation Offset Trajectory Description

Medial, Outboard Concept

The two terms 'inside' and 'outside' will be used in the following description '. When the angle between the intersection of two moving segments is greater than or equal to 180 °, it is called inboard. When the angle between 0 ° and 180 °, it is called outboard '.



Tool movement during starting

The realization of tool tip radius compensation involves three steps: tool compensation establishment, tool compensation, and tool compensation cancellation.

From the offset cancellation mode to the establishment of G41 or the G42 instruction, the movement of the tool is called the tool compensation establishment (also known as the starting tool).

Note: S, L and C marked in the following drawings have the following meanings unless otherwise specified:

S-single stop point; L-straight line; C- arc.

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

1) Straight Line-Straight Line

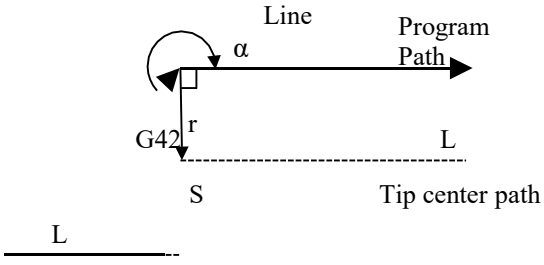


Fig. 4-14a Straight Line-Straight Line (Inner Knife)

2) Straight line-circle

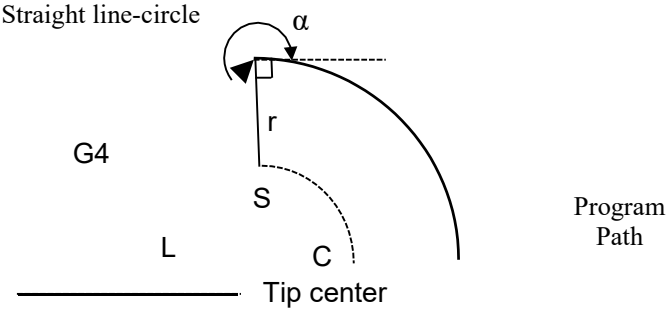


Fig. 4-14b Straight Line-Arc (Inner Knife)

(B) Move along the outer side of the obtuse corner (180 °>α≥ 90 °)

1) straight line-straight line

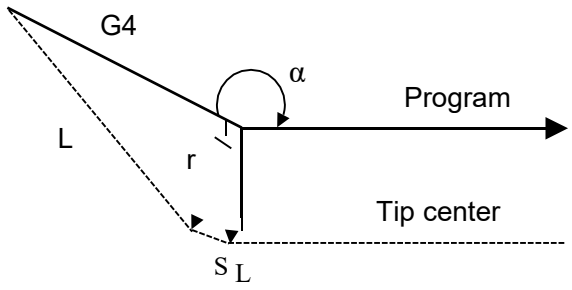


Fig. 4-15a Straight Line-Straight Line (Outside Trip)

2) straight line-arc

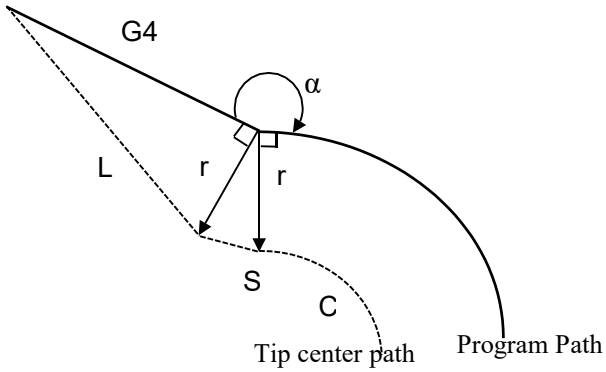


Figure 4-15b Straight Line-Arc (Outside Tight)

(c) Move along the outside where the corner is acute (α<90 °)

1) Straight Line-Straight Line

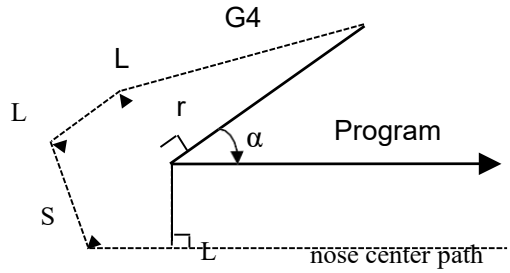


Fig. 4-16a straight line-straight line (outside cutting)

2) Straight line-arc

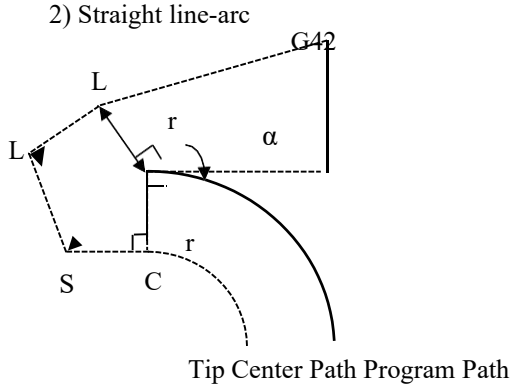


Fig. 4-16b straight line-arc (outside cutting)

(d) Move along the outside where the corner is an acute angle less than 1 degree, straight line→straight line. (α ≦ 1 °)

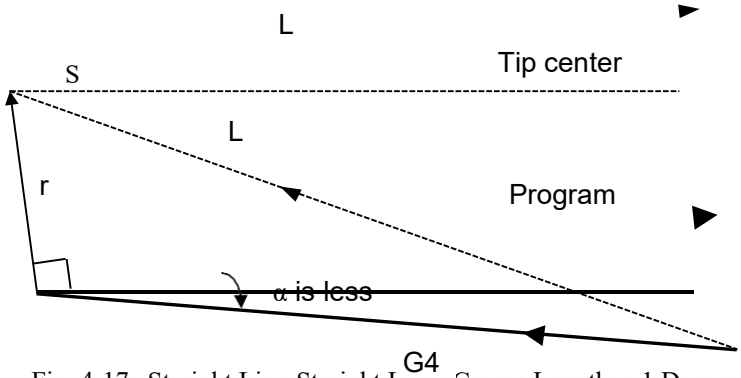


Fig. 4-17a Straight Line-Straight Line (Corner Less than 1 Degree, Outside Knives)

Tool movement in offset mode

After the tool tip radius compensation is established and before the tool tip radius compensation is canceled, it is called offset mode.

- Offset trace without changing compensation direction in compensation mode

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

1) Straight Line-Straight Line

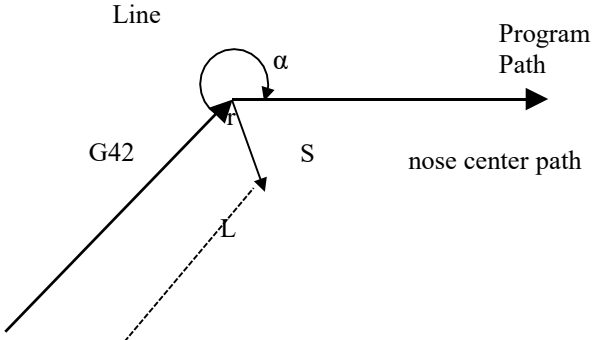


Fig. 4-18a Straight Line-Straight Line (Inner Movement)

2) Straight line-arc

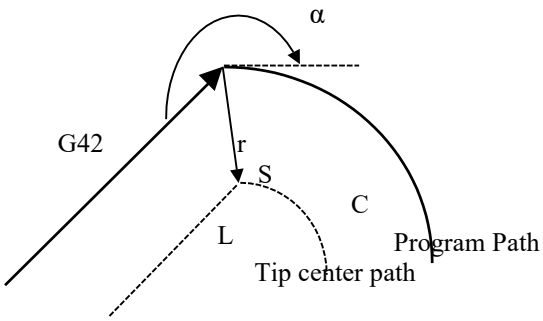


Figure 4-18b Line-Arc (Inner Movement)

3) Arc-straight line

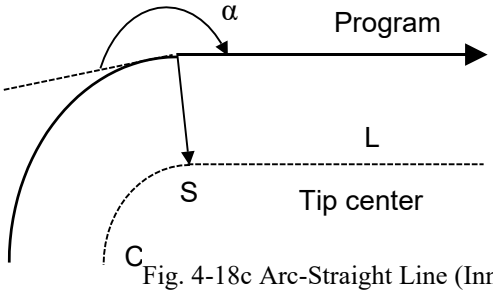


Fig. 4-18c Arc-Straight Line (Inner Movement)

4) Arc-arc

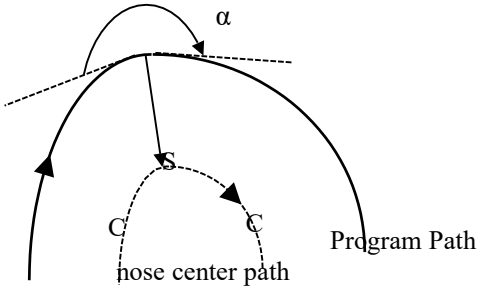


Fig. 4-18d Arc-Arc (Inner Movement)

5) Less than 1 degree inside processing and compensation vector amplification

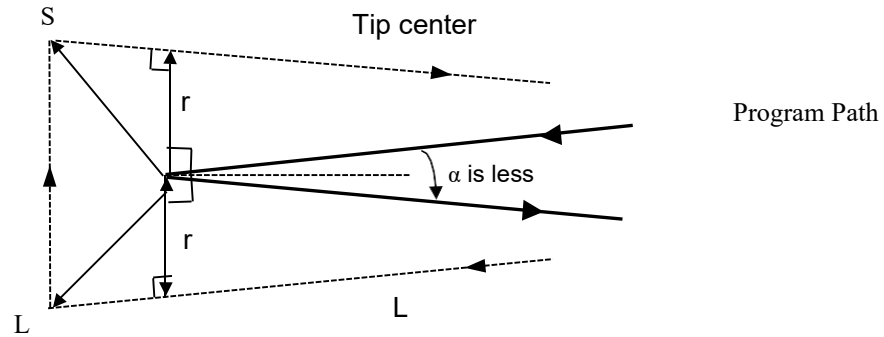


Figure 4-18e Straight line-straight line (corner less than 1 degree, inside movement)

(B) Move along the outer side of the obtuse corner (180 °>α≥ 90 °)

1) Straight
Line-Straight
Line

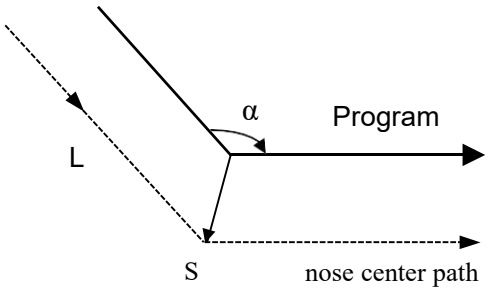


Fig. 4-19a straight line-straight line
(obtuse angle, lateral movement)

2) Straight line-arc

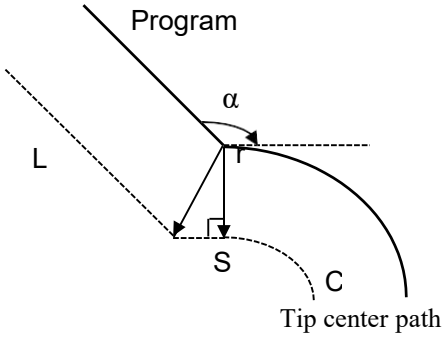


Fig. 4-19b straight line-arc
(obtuse angle, lateral movement)

3) Arc-straight
line

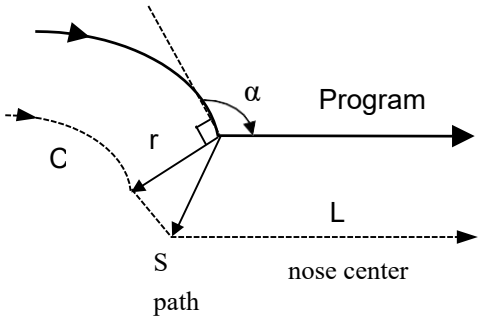


Fig. 4-19c arc-straight line
(obtuse angle, lateral movement)

4) Arc-arc

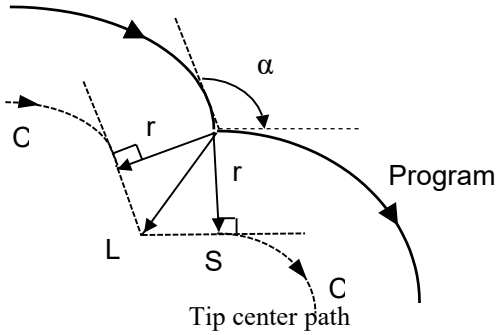


Fig. 4-19d arc-arc
(obtuse angle, lateral movement)

(c) Move along the outside where the corner is acute (α<90 °)

1) Straight
Line-Straight
Line

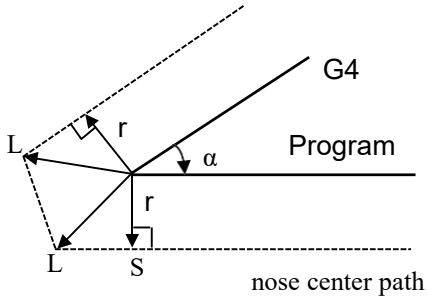


Fig. 4-20a straight line-straight line
(acute angle, lateral movement)

2) Straight line-arc

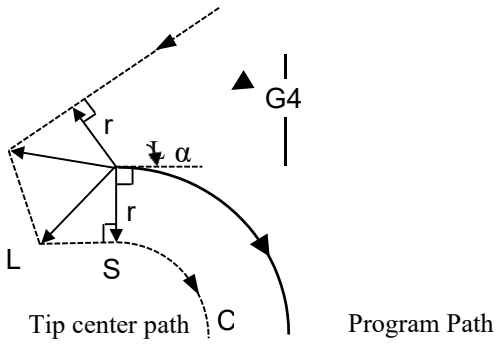


Fig. 4-20b straight line-arc
(acute angle, lateral movement)

3) Arc-straight line

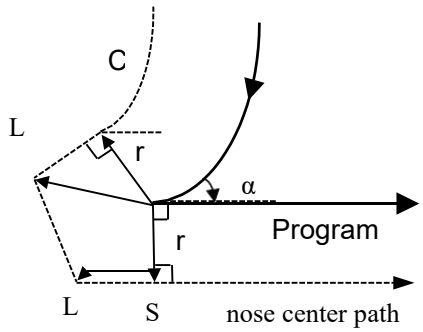


Fig. 4-20c arc-straight line
(acute angle, lateral movement)

Fig. 4-2 Arc-

Program Path

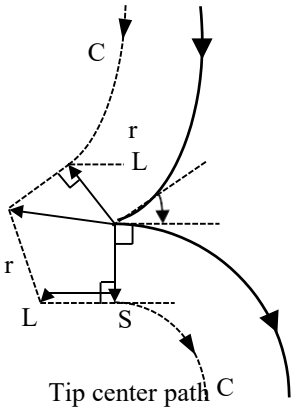
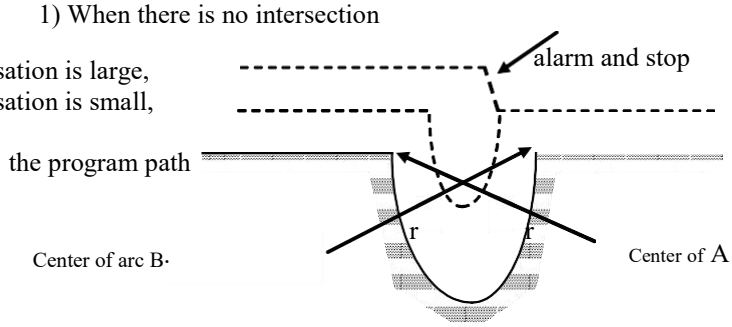


Fig. 4-20d arc-arc
(acute angle, lateral movement)

(d) Special circumstances

1) When there is no intersection

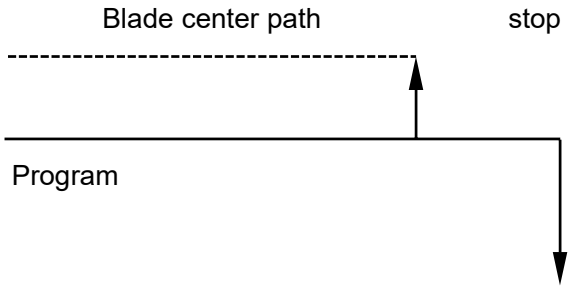
the amount of compensation is large,
the amount of compensation is small,



When the tool radius value is small, the compensation path of the arc has an intersection point, but when the radius becomes large, the intersection point may not exist. The tool stops in the previous program segment and displays the alarm.

Fig. 4-21 Special Case-Offset Trajectory Without Crossing

2) The center of the arc is consistent with the start or end point



generates an alarm: the starting point or end point of the arc is the same as the center of the circle; And stop at the end of the previous program segment

(G41)
N5 G01 W20;
N6 GO2 W10 I K0;
N7 G03 U-10 I-10;

Fig. 4-22 the center of the arc is consistent with the start point or end point

- Offset trace to change the compensation direction in compensation mode

The tool tip radius compensation G41 and G42 commands determine the compensation direction, and the symbol of the compensation amount is as follows

Table 4-2

Symbol of compensation amount / G code	+	-
G41	Left side compensation	Right compensation
G42	Right compensation	Left side compensation

In special cases, the compensation direction can be changed in the compensation mode. However, it is not possible to change the program segment at the beginning and the following program segment. When the compensation direction is changed, there is no concept of inside and outside for all situations. The following compensation amounts are assumed to be positive.

1) Straight Line-Straight Line

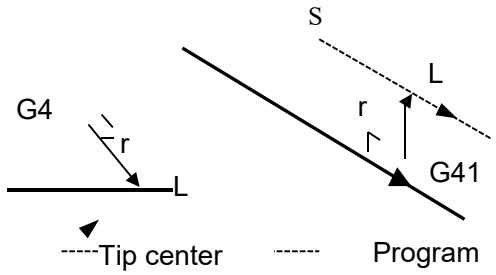


Figure 4-23 Straight Line-Straight Line (Change Compensation Direction)

2) Straight line-arc

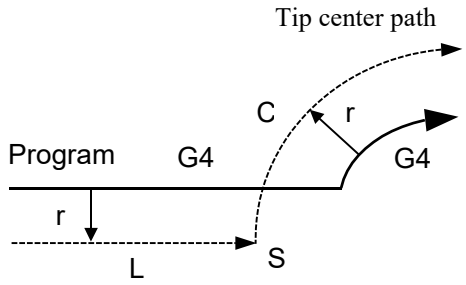


Figure 4-24 Line-Arc (Change Compensation Direction)

3) Arc-straight line

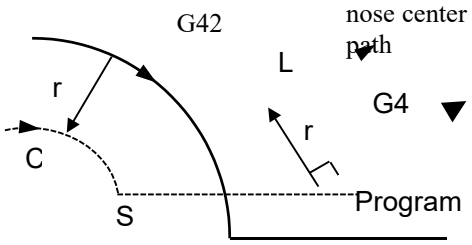


Fig. 4-25 Arc-Line (Change of Compensation Direction)

4) Arc-arc

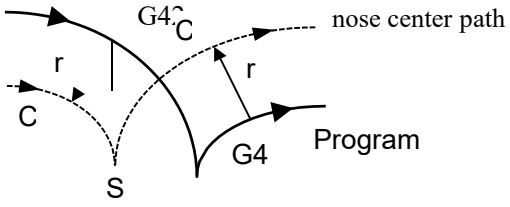


Figure 4-26 Arc-Arc (Change Compensation Direction)

5) If the compensation is executed normally, but there is no intersection

When G41 and G42 are used to change the bias direction of program segment A to program segment B, if the intersection of the bias path is not required, a vector perpendicular to program segment B is made at the beginning of program segment B.

i) Straight line ----
Straight line

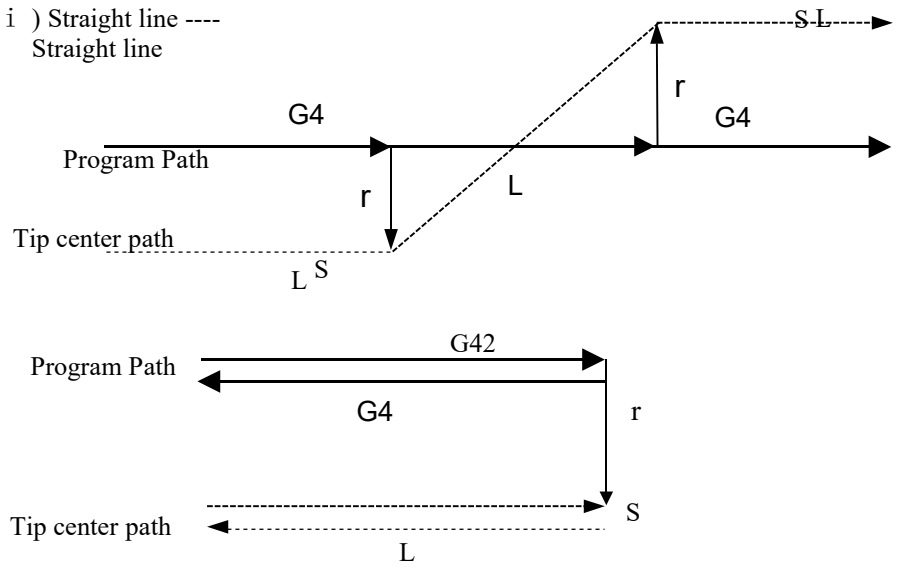


Fig. 4-27a Straight Line-Straight Line Without Intersection (Change of Compensation Direction)

ii) Straight line --- arc

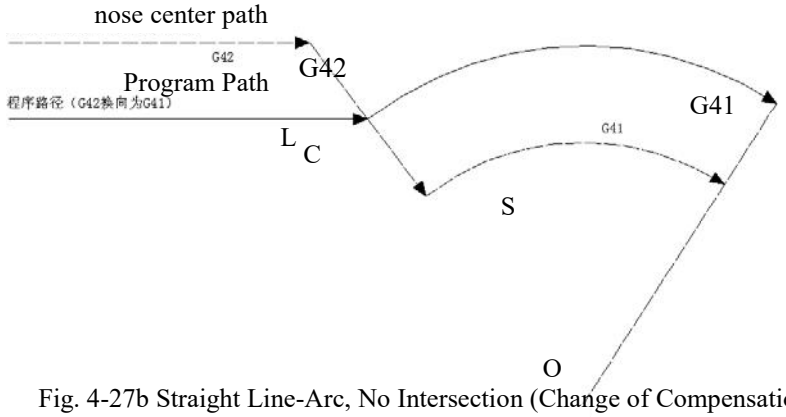


Fig. 4-27b Straight Line-Arc, No Intersection (Change of Compensation Direction)

iii) Arc ----- Arc

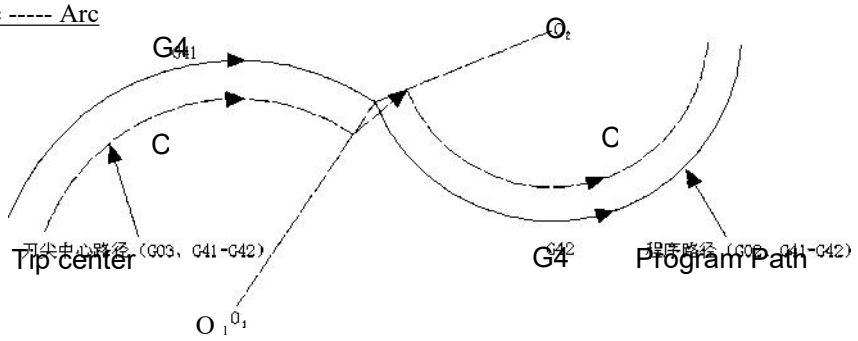


Fig. 4-27c Arc-Arc, No Intersection (Change of Compensation Direction)

Tool movement in offset cancellation mode

In the compensation mode, when the program segment meets any of the following conditions, the CNC enters the compensation cancellation mode, and the action of this program segment is called compensation cancellation.

1, the use of G40 instructions in the program;

2. Execute the M30 instruction.

When the C cutter compensation is canceled, the arc command (G02 and G03) is not available. If

the command arc generates an alarm (N0.34) and the operation stops. In the compensation cancellation mode, control executes the program segment and the program segment in the nose radius compensation buffer register. At this point, if the single program segment is open.

When OFF is ON, stop after executing a program segment. Press the start button again to execute the next program segment without reading the next program segment.

After the control is in cancel mode, usually the next program segment to be executed will be read into the buffer register, and the next program segment will not be read into the tool tip radius compensation buffer.

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

1) Straight line → Straight line

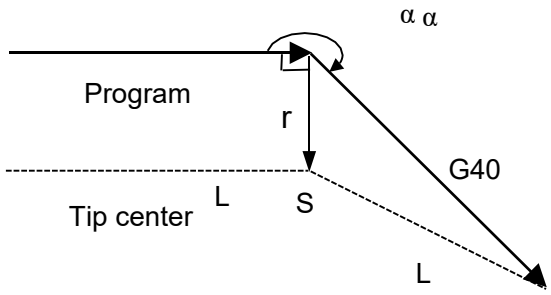


Fig. 4-28a Straight Line-Straight Line (Inside, Offset Cancelled)

2) Arc → Line

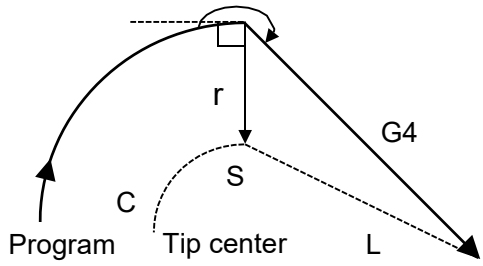


Fig. 4-28b Arc-Straight Line (Inside, Offset Cancelled)

(B) Move along the outer side of the obtuse corner (180 °>α≥ 90 °)

1) Straight line →
Straight line

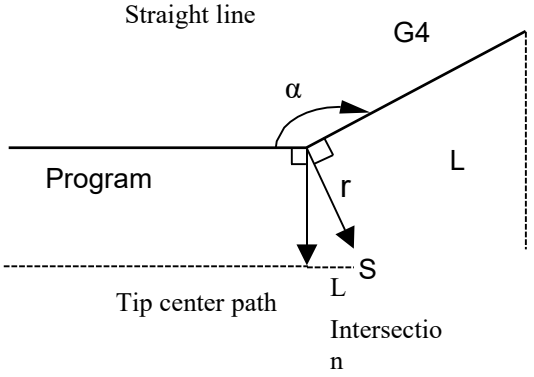


Fig. 4-29a Straight Line-Straight Line
(Obtuse Angle, Outside, Unoffset)

2) Arc → Line

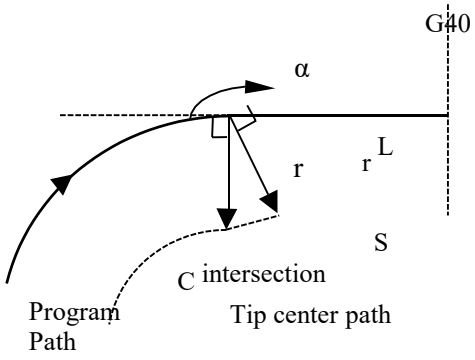


Fig. 4-29b Arc-Straight Line
(Obtuse Angle, Outside, Unoffset)

(c) Move along the outside where the corner is acute (α<90 °)

1) Straight line → Straight line

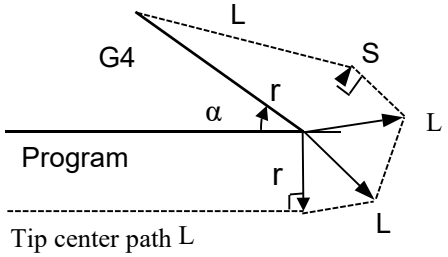


Fig. 4-30a Straight Line-Straight Line
(Acute Angle, Outside, Offset Cancellation)

2) arc → straight line

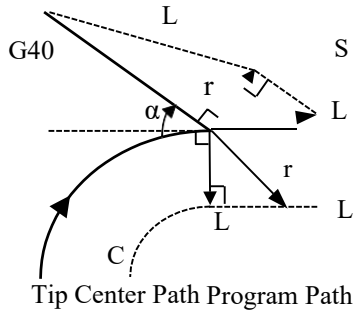


Fig. 4-30b Straight Line-Straight Line
(Acute Angle, Outside, Offset Cancellation)

(D) Move along the outside where the corner is an acute angle less than 1 degree; straight line → straight line. (α<1 °)

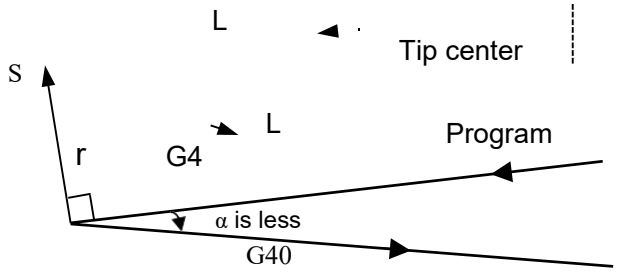


Figure 4-31 Straight line-straight line (included angle less than 1 degree, outside, offset canceled)

Tool interference check

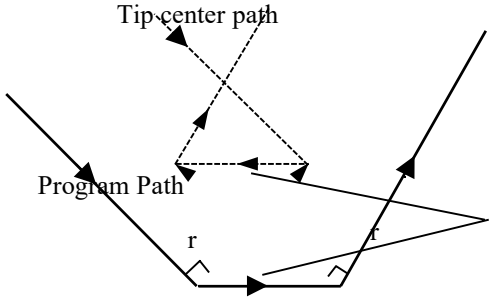
Tool transition cutting is called "interference". Interference can check tool transition cutting in advance, and interference check will be carried out even if transition cutting does not occur. But not all tool interference can be detected.

（1） basic conditions of interference

1） The tool path direction is different from the program path direction. (The angle between the paths is between 90 ° and 270 °).

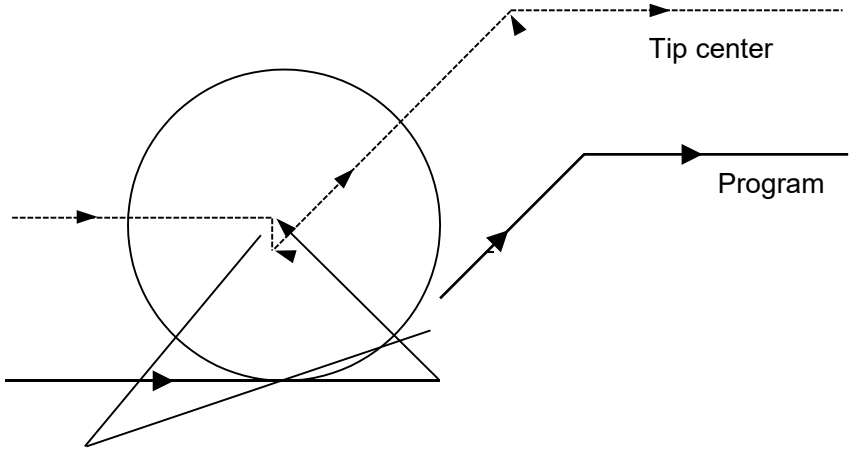
2） In addition to the above conditions, the angle between the starting point and the end point of the tool center path is very different from the angle between the starting point and the end point of the program path (more than 180 °).

Example: Straight line machining



two directions is very different (180 °)

Fig. 4-32a Processing Interference (1)



The direction of the two paths is very different (180 °)

Fig. 4-32b Processing Interference (2)

(2) There is actually no interference, and it is also treated as interference.

1) Groove depth less than compensation amount

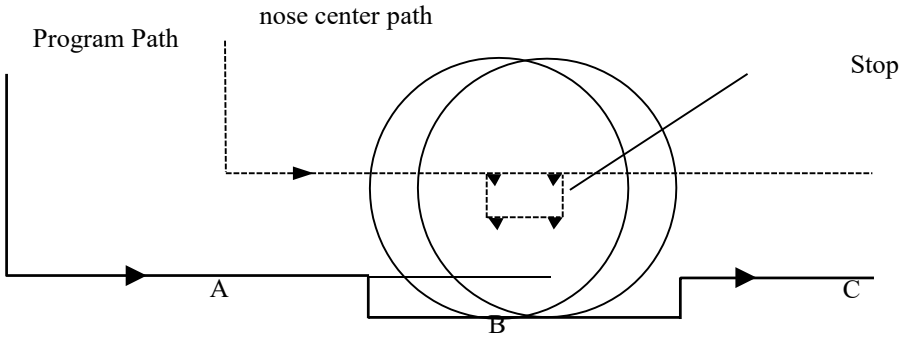


Figure 4-33 Interference Handling Special Case (1)

There is actually no interference, but in segment B the direction of the program is opposite to the path of the nose radius compensation, the tool stops and an alarm is displayed.

2) Groove depth less than compensation amount

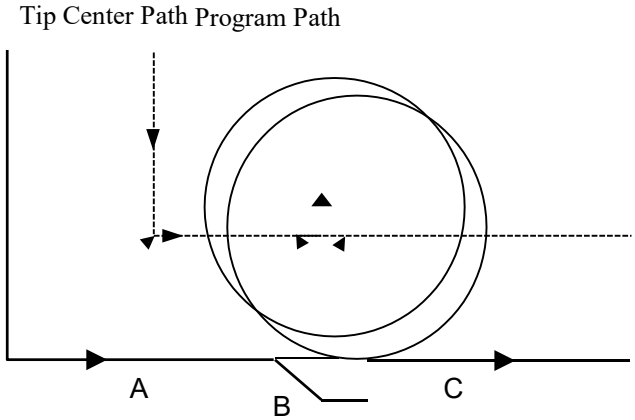


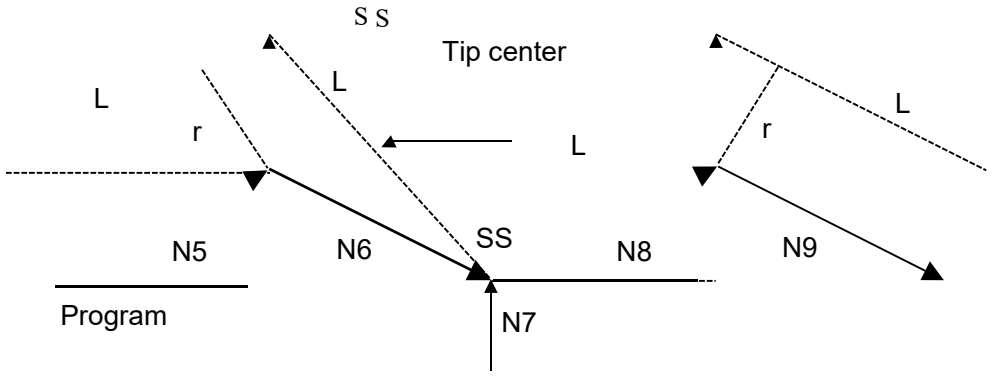
Figure 4-34 Several special cases for interference processing (2)

There is actually no interference, but in segment B the direction of the program is opposite to the path of the nose radius compensation, the tool stops and an alarm is displayed.

Temporarily cancel the instruction to compensate the vector

In the compensation mode, if G50, G71 ~ G76 instructions are specified, the compensation vector will be temporarily canceled, and the compensation vector will be automatically restored after the instruction is executed. The compensation temporary cancellation at this time is different from the compensation cancellation mode in that the tool moves directly from the intersection point to the instruction point of compensation vector cancellation. When the compensation mode is restored, the tool moves directly to the intersection point again.

- Coordinate System Setting **G50** Commands



G50 Program Segment

Figure 4-35 G50 instruction temporary compensation vector

Note: SS indicates the point where the tool stops twice in the single program segment mode.

- G28 Automatic return to reference point

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

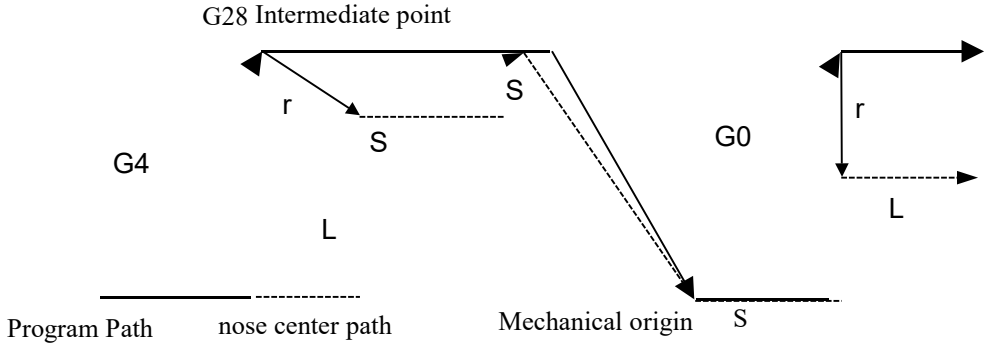


Figure 4-36 G28 Temporarily Cancels Compensation Vector

- **G71** ~ **G76** compound cycle; **G32**, **G33**, **G34** thread cutting.

When G71 ~ G76 fixed cycle instructions are executed; G32, G33, G34 thread cutting instructions, in the cycle process, do not perform nose radius compensation, temporarily cancel the nose radius compensation, in the following program segment G00, G01, G02, G03, G70 instructions, CNC will compensate the mode automatically restored.

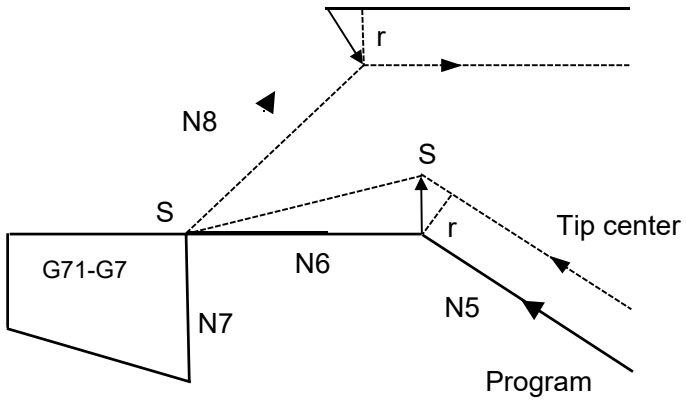


Figure 4-37 G71 ~ G76 Temporarily Cancel Compensation Vector

- **G90**, **G94**

code instruction G90 or G94 code instruction to perform tool tip

radius compensation:

A. For each path of the loop, the nose center path is parallel to the program path.

B. Using G41 and G42 instructions, the bias direction is the same, as shown in the figure below.

C. The is compensated according to the fixed imaginary tool tip direction No. 0 (for example, the imaginary tool tip direction T1 ~ T9 is also compensated according to No. T0), the running track is offset from the tool tip radius vector, and no intersection calculation is performed during the cycle.

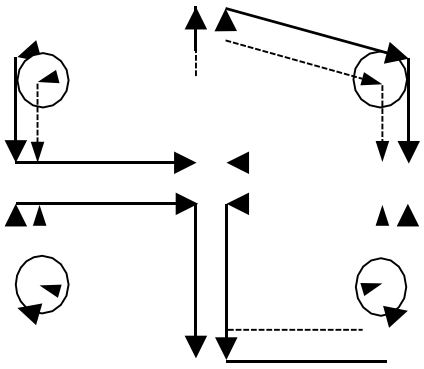


Fig. 4-38 Offset Direction of G90 Tip Radius Compensation

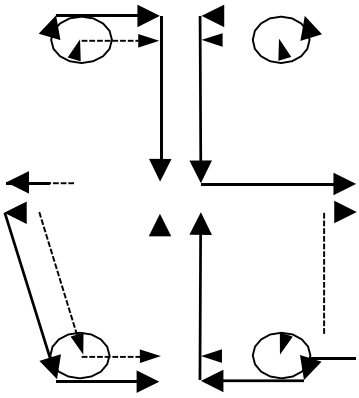


Figure 4-39 Offset Direction for G94 Narrow Radius Compensation

Special circumstances

- When the inner corner is machined less than the nose radius

At this time, the inside offset of the tool will cause excessive cutting. At the beginning of the previous program segment or after the corner moves, the tool movement stops and the alarm is displayed

(P/S41). However, if the 'Single Segment' switch is ON, the tool will stop at the end of the previous segment.

- When machining a concave shape smaller than the nose diameter

When the nose radius compensation makes the center of the nose move in the opposite direction to the program path, overcutting will occur. At this time, after the beginning of the previous program segment or the corner movement, the tool movement stops and an alarm is displayed.

- When machining a step smaller than the nose radius

When the program contains a step smaller than the nose radius and the step is a circular arc, the tool center path may form a direction of motion opposite to the program path. This automatically ignores the first vector and moves straight to the end of the second vector. In the single program segment, the program will stop at this point. If it is not in the single program segment mode, the loop operation will continue. If the step is straight, the compensation will be performed correctly without generating an alarm. (However, the uncut part remains)

- When subroutine is included in G code

Before calling the subroutine (I. e. before executing M98), the CNC must be in compensation cancellation mode. After entering the subroutine, the offset can be started, but it must be in the compensation cancellation mode before returning to the main program (I. e. before executing M99). Otherwise, an alarm will appear.

- When the compensation amount is changed

Usually, the value of the compensation amount is changed when the mode tool change is canceled. If the compensation amount is changed in the compensation mode, the new compensation amount is effective only after the tool change.

Positive and negative compensation and nose center path

If the compensation amount is negative (-), G41 and G42 are interchanged with each other in the program. If the tool center moves along the outside of the workpiece, it will move along the inside, and vice versa.

Generally, the amount of compensation during the production process is (+). When the tool path is made as shown in (a), if the compensation amount is taken as negative (-), the tool center moves as shown in (B), and vice versa.

Also note that when the offset sign changes, the nose offset direction also changes, but the imaginary nose direction does not change. So do not arbitrarily change the sign of the offset.

The end of the programmed arc is not on the arc

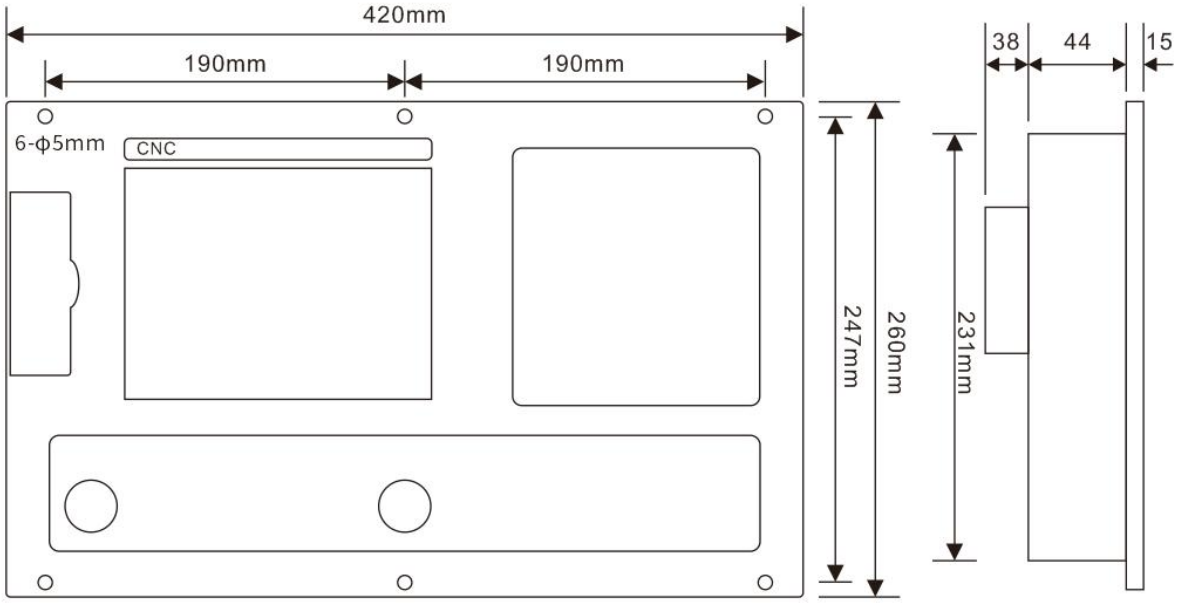
When the end of the arc in the program is not on the arc, the tool motion stops and the alarm message "the end of the arc is not on the arc" is displayed.

System installation dimensions

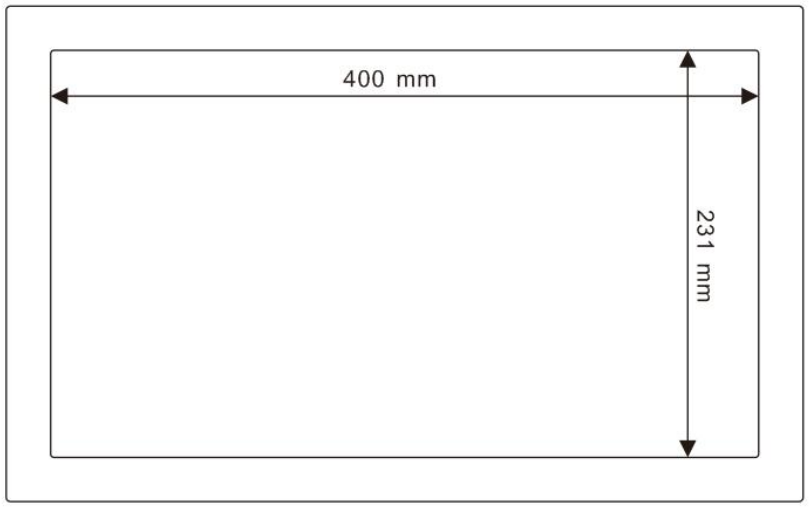
Note: The switching power supply L and N must be connected to AC 220V and current 0.5A through isolation transformer.

1. main panel: NG98T, NG98Ta, NG98TT panel size is as follows:

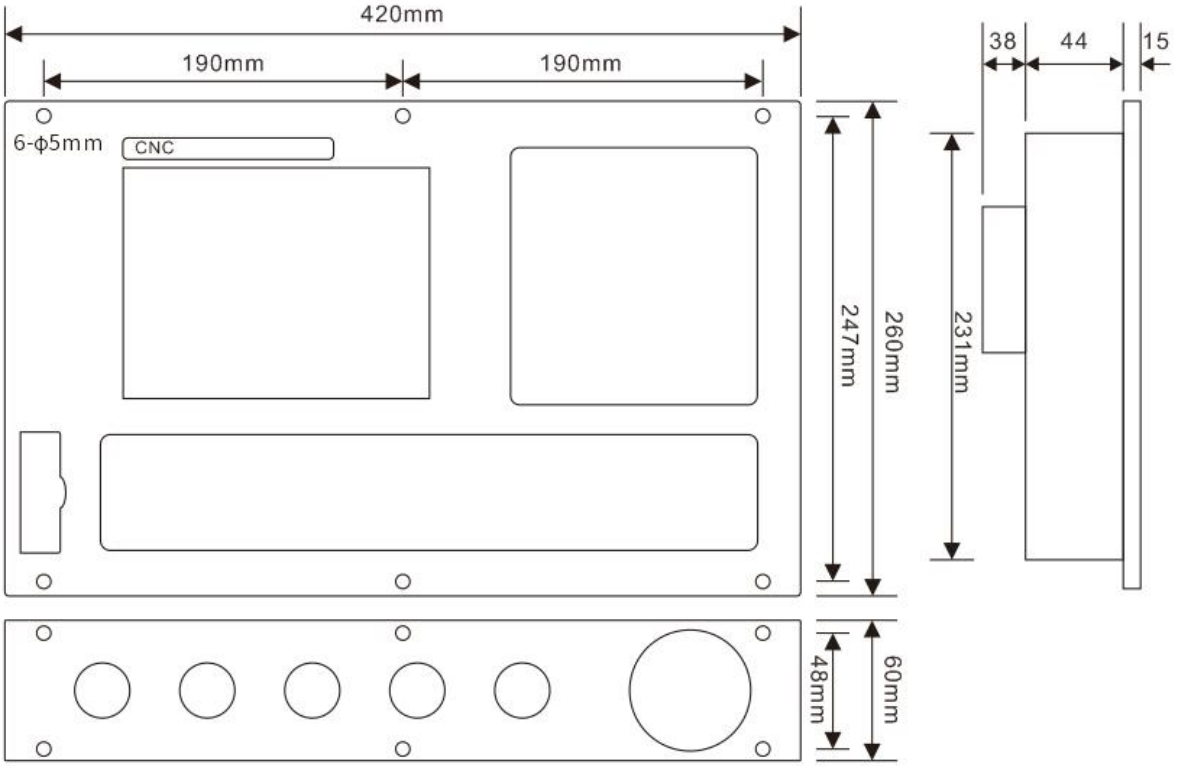
1. NG98Ta system dimensions



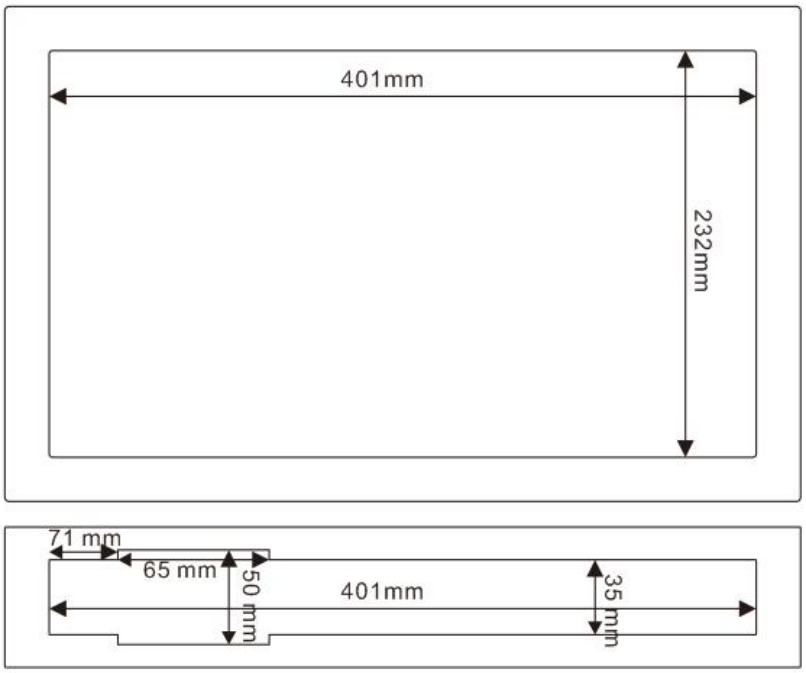
系统后盖尺寸：



2, NG98T, NG98TT system size

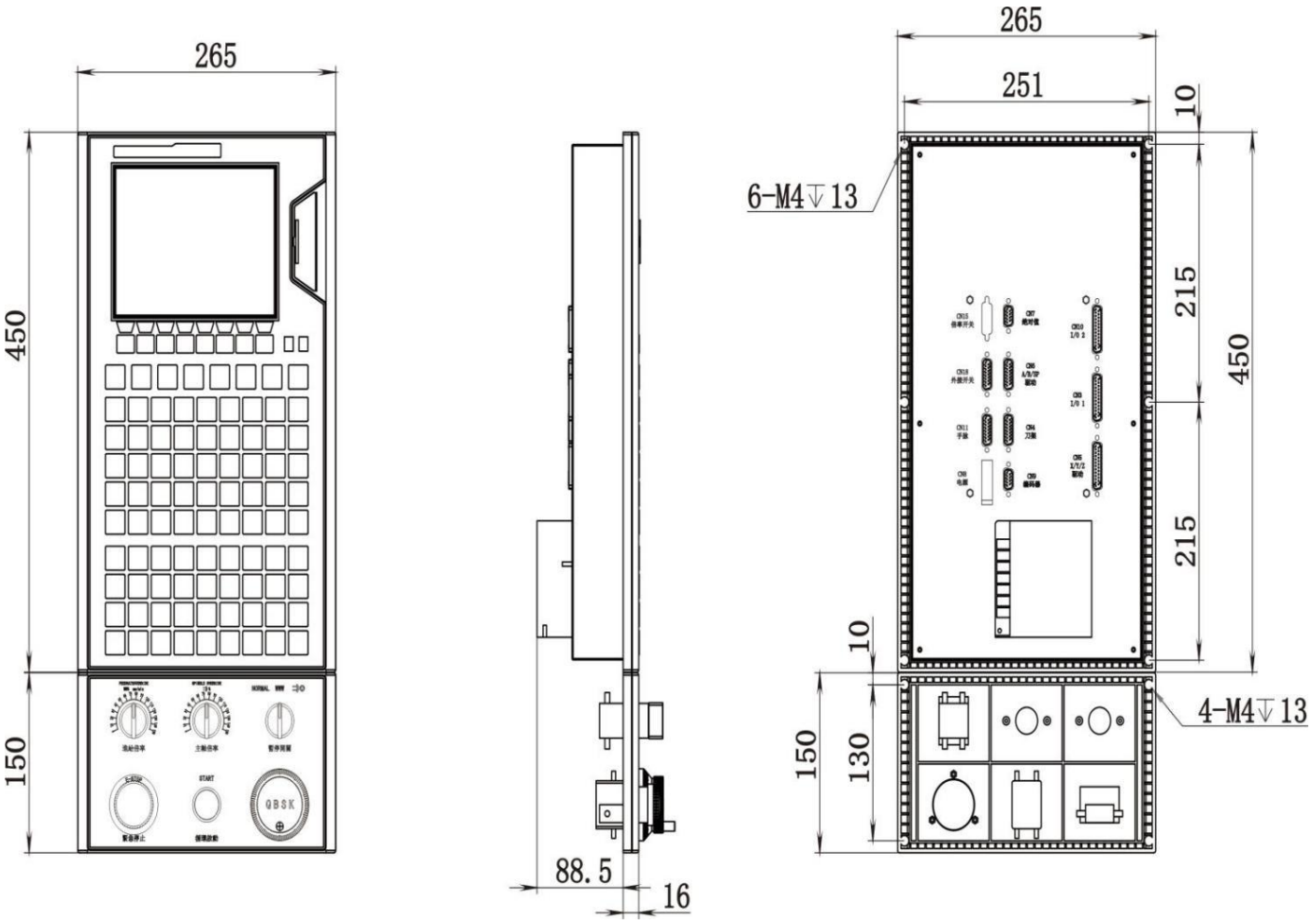


系统后盖尺寸：

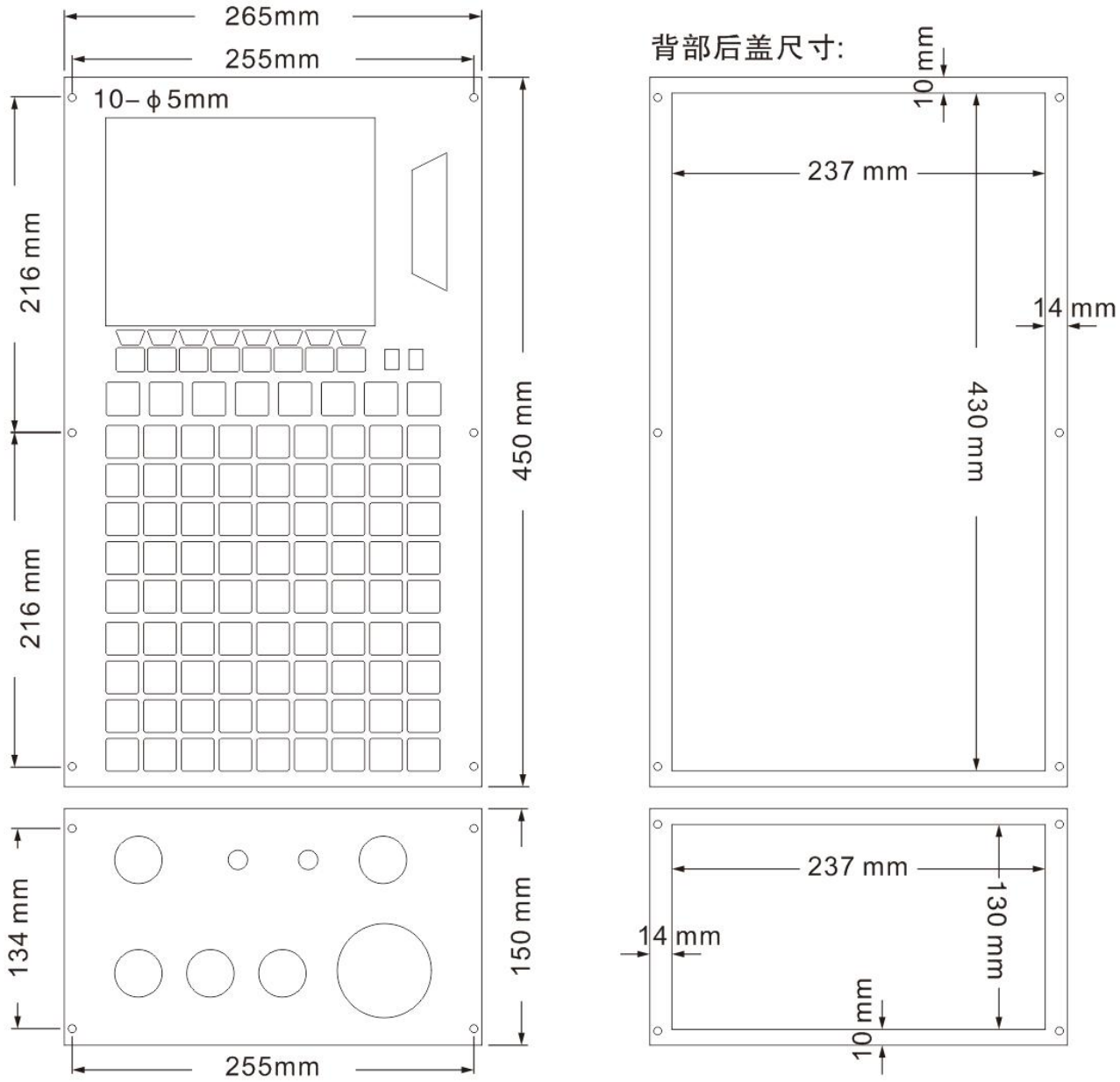


2. main panel: NG98T-V, NG98TT-V

panel size 1、Back mounting method:



2 and front installation methods



System rear interface connection diagram

一、Connection DB15 hole holder of electric tool holder CN4

	CN4 electric tool holder signal DB15 hole holder				
Signal	Foot	I /O	Function	Signal bit	effective
0V	10	OUT	Power ground		0V
+24V	11, 15	OUT	24 Power		+24V
T+	12	OUT	Tool holder forward rotation	O10	0V
T-	13	OUT	Tool holder reversal signal	O11	0V
T1	1	IN	No. 1 tool position signal	I08	0V
T2	2	IN	No. 2 tool position signal	I09	0V
T3	3	IN	No. 3 tool position signal	I10	0V
T4	4	IN	No. 4 tool position signal	I11	0V
T5	5	IN	No. 5 Tool Signal	I12	0V
T6	6	IN	No. 6 tool position signal	I13	0V
T7	7	IN	No. 7 tool position signal	I14	0V
T8	8	IN	No. 8 tool position signal	I15	0V
TOK	9	IN	Tool holder lock signal	I21	0V

This system can control 1-32 knives, of which the electric knife holder is up to 8, and the default is 4. .

Comprehensive parameter No. 4: electric tool holder function, you can choose "install electric tool holder" or "do not install",

台湾六鑫刀塔接线图				
数控系统信号	接口脚	六鑫刀架功能	功能	有效电平
T1	CN4-1	A信号	输入信号	0V
T2	CN4-2	A信号	输入信号	0V
T3	CN4-3	A信号	输入信号	0V
T4	CN4-4	A信号	输入信号	0V
T5	CN4-5	数刀信号	输入信号	0V
T6	CN4-6	锁紧到位检测	输入信号	0V
M59	CN10-6	锁紧输出信号	输出信号	0V
M61	CN10-1	松开输出信号	输出信号	0V
T+	CN4-12	刀架正转输出信	输出信号	0V
T-	CN4-13	刀架反转输出信	输出信号	0V

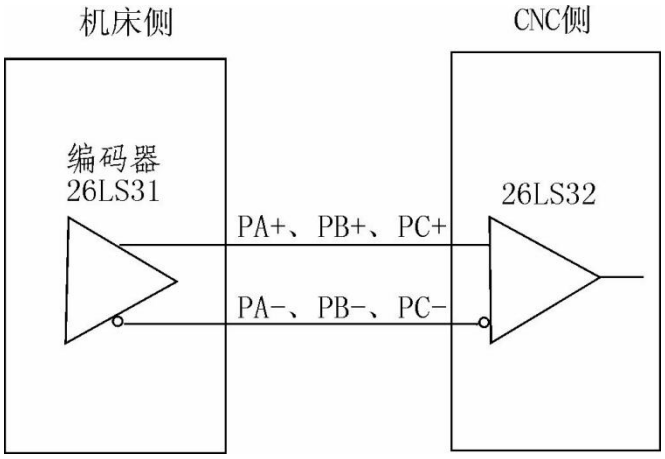
二、 spindle encoder socket CN9 connection DB9 pin socket

CN9 spindle encoder signal DB9 needle holder				
Signal	Foot No.	I /O	Function	effective level
0V	4	OUT	Power ground	0V
+5V	1	OUT	5V power supply	+5V
PA+	5	IN	A signal is positive	5V
PA-	7	IN	A signal negative	
PB+	3	IN	B signal positive	5V
PB-	6	IN	B signal negative	
PC+	2	IN	Z sync signal positive	5V
PC-	8	IN	Z sync signal negative	

Note: 1. The encoder output signal adopts long-line output mode, and the power supply is 5V.

2、The signal line must be a twisted pair shielded cable with a length of no more than 20m.

Encoder input signal PA, PB, PC schematic diagram:



三、 input and output CN3 connection DB25 hole seat

CN3 I/O1 machine signal DB25 hole holder					
Signal	Foot No.	I /O	Function	Signal bit	effective
0V	1	OUT	Power ground		0V
+24V	14	OUT	24 Power		+24V
Y0	2	IN	Y-axis zero return signal	I27	0V
X0	3	IN	X-axis zero return signal	I28	0V
Z0	17	IN	Z-axis zero return signal	I29	0V
L-	15	IN	negative limit signal of machine tool	I25	0V
L+	16	IN	Positive limit signal of machine tool	I26	0V
A0	4	IN	A- axis zero return signal	I24	0V
ALM1	5	IN	Machine Converter Alarm 1	I32	0V
HALT/SP	6	IN	External pause button signal	I31	0V
RUN/ST	18	IN	External operation button signal	I30	0V
M03	19	OUT	spindle forward rotation	O06	0V
M04	7	OUT	Spindle reversal	O05	0V
M05	20	OUT	Spindle stop	O04	0V
M08	8	OUT	Cooling	O03	0V
M10	21	OUT	Spindle chuck	O02	0V
M32	9	OUT	machine tool lubrication	O00	0V
M79	22	OUT	spindle thimble	O01	0V
M41	10	OUT	Spindle 1st gear	O15	0V
M42	23	OUT	Spindle 2nd gear	O14	0V
M43	11	OUT	Spindle 3rd gear	O13	0V
M44	24	OUT	Spindle 4th gear	O012	0V
M75	12	OUT	Output signal	O07	0V
+10V	25	OUT	First spindle frequency conversion		0 to 10V
0V	13	OUT	Frequency conversion ground		0V

四、input and output and connection DB25 hole seat of C axis CN10 Connection DB26 socket for

	CN10 I/O2 machine signal DB25 hole seat				
Signal	Foot No.	I /O	Function	Signal bit	effective level
0V	1	OUT	Power ground		0V
+24V	14	OUT	24 Power		+24V
CCP+	15	OUT	C- axis pulse signal positive		0V
CCP-	16	OUT	C- axis pulse signal negative		
CDIR+	17	OUT	C- axis direction signal positive		0V
CDIR-	18	OUT	C- axis direction signal negative		
ALM	4	IN	Drive alarm	I22	0V
ALM2	2	IN	Machine Alarm 2	I23	0V
B0	3	IN	B- axis zero return signal	I34	0V
C0	5	IN	C- axis zero return signal	I35	0V
M59	6	OUT	Custom Output	O31	0V
M61	19	OUT	Custom Output	O24	0V
M63	7	OUT	Custom Output	O25	0V
M65	20	OUT	Three-color lamp stop signal	O26	0V
M67	8	OUT	Three-color lamp emergency stop signal	O27	0V
M69	21	OUT	Three-color lamp operation signal	O28	0V
M11	9	OUT	Chuck release signal	O29	0V
M78	22	OUT	Tailstock release signal	O30	0V
M18	10	IN	Custom Input	I36	0V
M28	23	IN	Custom Input	I33	0V
M12	11	IN	Door switch input signal	I37	0V
M14	24	IN	External tailstock start input signal	I38	0V
M16	12	IN	External chuck start input signal	I39	0V
+10V	25	OUT	Second spindle frequency conversion		0 to 10V
0V	13	OUT	Frequency conversion ground		0V

五、input and output CN16 (NG98T, NG98Ta and NG98TT do not have this interface)

	CN16 I/O3 machine signal DB26 hole holder				
Signal	Foot No.	I /O	Function	Signal bit	effective level
0V	20, 21	OUT	Power ground		0V
+24V	10, 19	OUT	24 Power		+24V
X40	1	IN	Alternate input 0	I48	0V
X41	2	IN	Alternate Input 1	I49	0V
X42	3	IN	Alternate Input 2	I50	0V
X43	4	IN	Alternate Input 3	I51	0V
X44	5	IN	Alternate Input 4	I52	0V
X45	6	IN	Alternate Input 5	I53	0V
X46	7	IN	Alternate Input 6	I54	0V
X47	8	IN	Alternate Input 7	I55	0V
Y24	11	OUT	Standby output 0	O16	0V
Y25	12	OUT	Standby output 1	O17	0V
Y26	13	OUT	Standby output 2	O18	0V
Y27	14	OUT	Standby output 3	O19	0V
Y28	15	OUT	Standby output 4	O20	0V
Y29	16	OUT	Standby output 5	O21	0V
Y30	17	OUT	Standby output 6	O22	0V
Y31	18	OUT	Standby output 7	O23	0V
RUN/ST	24	IN	External operation	I30	0V
HALT/SP	23	IN	External stop	I31	0V

六、X/Y/Z drive signal socket CN5 connection DB25 pin socket

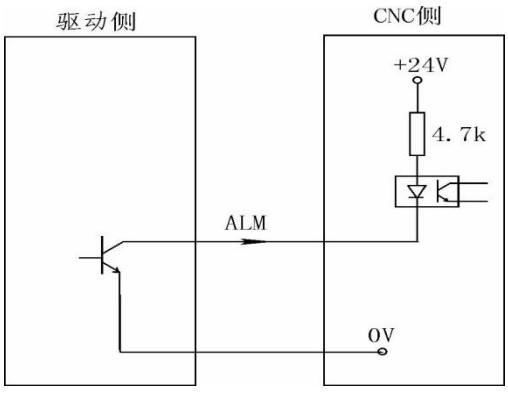
CN5 servo drive signal DB25 needle holder				
Signal	Foot No.	I /O	Function	effective
XCP+	6	OUT	X-axis pulse signal positive	5V
XCP-	18	OUT	X-axis pulse signal negative	
XDIR+	7	OUT	X-axis direction signal positive	5V
XDIR-	19	OUT	X-axis direction signal negative	
YCP+	8	OUT	Y-axis pulse signal positive	5V
YCP-	20	OUT	Y-axis pulse signal negative	
YDIR+	9	OUT	Y-axis direction signal positive	5V
YDIR-	21	OUT	Y-axis direction signal negative	
XZO+	5	IN	X-axis motor zero positive	5V
XZO-	17	IN	X-axis motor zero negative	
ZCP+	3	OUT	Z-axis pulse signal positive	5V
ZCP-	15	OUT	Z-axis pulse signal negative	
ZDIR+	4	OUT	Z-axis direction signal positive	5V
ZDIR-	16	OUT	Z-axis direction signal negative	
ZZO+	2	IN	Z-axis motor zero positive	5V
ZZO-	14	IN	Z-axis motor zero negative	
YZ0 +	24	IN	Y-axis motor zero positive	5V
YZ0-	25	IN	Y axis motor zero negative	
0V	13, 23	OUT	Power ground	0V
ALM	12	IN	Servo alarm	0V
+24V	11	OUT	+24V power supply	24V

七、 the connection DB15 socket of the and B axis drive signal socket CN6

CN6 servo drive signal DB15 hole holder				
Signal	Foot No.	I /O	Function	effective level
ACP+	1	OUT	A- axis pulse signal positive	5V
ACP-	9	OUT	A- axis pulse signal negative	
ADIR+	2	OUT	A- axis direction signal positive	5V
ADIR-	10	OUT	A- axis direction signal negative	
BCP+	4	OUT	B- axis pulse signal positive	5V
BCP-	12	OUT	B- axis pulse signal negative	
BDIR+	5	OUT	B- axis direction signal positive	5V
BDIR-	13	OUT	B- axis direction signal negative	
AZO+	3	IN	A- axis motor zero positive	5V
AZO-	11	IN	A- axis motor zero negative	
BZO+	6	IN	B- axis motor zero positive	5V
BZO-	14	IN	B- axis motor zero negative	
0V	8	OUT	Power ground	0V
ALM	7	IN	Servo alarm	0V
+24V	15	OUT	+24V power supply	24V

Note: 1. The twisted pair shielded cable must be used for the drive signal line, and the length shall not exceed 20m.
2. The input mode of alarm signal ALM normally open and normally closed is determined by the D0 bit parameter No. 31 of "comprehensive parameter.

servo alarm signal schematic diagram:



八、 electronic handwheel socket CN11 connection DB15 pin holder

		CN11 Electronic handwheel signal DB15 needle holder			
Signal	Foot No.	I /O	Function	Signal bit number	effective level
0V	13	OUT	Power ground		
+5V	6	OUT	5V power supply		+5V
PA+	8	IN	A signal is positive		0V
PA-	15	IN	A signal negative		
PB+	7	IN	B signal positive		0V
PB-	14	IN	B signal negative		0V
STOP	5	IN	External emergency stop signal	I20	0V
B	12	IN	B- axis select signal		0V
X100	4	IN	100 gear signal	I06	0V
X10	11	IN	10 Gear signal	I05	0V
X1	3	IN	1 Gear signal	I04	0V
A	10	IN	A- axis select signal	I03	0V
Z	2	IN	Z-axis selection signal	I02	0V
Y	9	IN	Y-axis select signal	I01	0V
X	1	IN	X-axis select signal	I00	0V

Note: All signals are internally supplied with a +5V power supply, so do not access a voltage higher than 5V.

九、Connection of magnification switch CN15 DB15 pin holder

Signal	CN15 pin holder	I /O	Signal bit number	Function
0V	2, 6	OUT		D signal
+5V	9, 10, 13	OUT		5V power
DS0	12	IN	I43	A signal
DS1	5	IN	I42	F signal
DS2	14	IN	I41	B signal
DS3	7	IN	I40	E signal
DK0	8	IN	I47	A signal
DK1	15	IN	I46	F signal
DK2	4	IN	I45	B signal
DK3	11	IN	I44	E signal

When the "comprehensive parameter" parameter 189 and parameter 190 are both set to "1", the band switch can be used. DS0, DS1, DS2 and DS3 input signals for the main shaft speed adjustment gear, and the 4 position controls a total of 16 gears. DK0, DK1, DK2 and DK3 are input signals for the feed speed adjustment gear, and the 4 position controls a total of 16 gears.
Note: All signals are internally supplied with a +5V power supply, so do not access a voltage higher than 5V.

External run, stop button

When the 185 number of the "comprehensive parameter" is set to "1", the 18 pins of CN3 can be used as the external operation button control program to automatically run; When the 186 number of the comprehensive parameter is set to "1", the 6 pins of CN3 can be used as the external stop button control program to pause.

Use of external emergency stop button

The ES signal is the input signal of the external emergency stop button, and the parameter No. 188 of "comprehensive parameter" can select whether the external emergency stop switch is normally open or normally closed.

十、Connection DB9 socket of absolute value interface CN7

	CN7 absolute value interface DB9 hole seat				
Signal	Foot	I /O	Function	Signal bit	effective
RS485-	1	IN	RS485-		
+24V	2	OUT	24V power		+24V
GND	3	OUT	Power		0V
X41	4	IN	Input	I49	0V
X40	5	IN	Input	I48	0V
RS485 +	6	IN	RS485 +		
Y24	7	OUT	Output	O16	0V
Y25	8	OUT	Output	O17	0V
Y26	9	OUT	Output	O18	0V

十一、998T, NG98Ta, NG98T-V external switch CN18 connection DB15 hole seat

CN18 external switch DB15 hole seat			
Signal	Foot No.	Function	effective level
RIGHI	1	Pause screw spindle signal	0V
RUN	9	Start signal	0V
STOP	14	Emergency stop signal	0V
LEFT	2	Pause screw signal	0V
0V	7	24V ground	0V
	8		0V
	13		0V
	15		0V

十二、998TT, NG98TT-V external switch CN18 connection DB15 hole seat

CN18 external switch DB15 hole seat			
Signal	Foot No.	Function	effective level
1 CHANNEL RIGHI	4	1 channel pause screw spindle signal	0V
1 channel RUN	10	1 channel start signal	0V
STOP	14	3 channels total emergency stop signal	0V
1 CHANNEL LEFT	12	Channel 1 pause screw signal	0V
2 CHANNEL RIGHI	3	2-channel pause screw spindle signal	0V
2-channel RUN	6	2 channel start signal	0V
2-CHANNEL LEFT	11	2-channel pause screw signal	0V
3 CHANNEL RIGHI	1	3-channel pause screw spindle signal	0V
3-channel RUN	9	3 channel start signal	0V
3 CHANNEL LEFT	2	3-channel pause screw signal	0V
0V	7	24V ground	0V
	8		0V
	13		0V
	15		0V
Total RUN	5	6, 9, 10 pin parallel output	0V

If you need to start RUN of 3 channels in total, please connect pins 6, 9 and 10 of CN18 in parallel to pin 5 output

System with 485 absolute motor adjustment method as follows:
1, open the power supply;
2. Setting system: whether the setting of comprehensive parameter no. 194 is equipped with absolute values x, y, z, a, B, C
3. 3. Setting system: whether the soft limit of comprehensive parameter No. 187 is effective; setting the ratio of electronic gears of each shaft; Then the direction signals of each shaft motor are arranged.
4. Setting synthetic parameter 195 = 92; 196 = 91; 197 = 90 absolute encoder multiple turn number 198 = 131072; synthetic parameter 199-204 setting coordinates for each turn (in NM) , for example: Z-axis straight screw pitch of 10 mm, 201 parameter = 1000000, for example: Z-axis deceleration ratio of 2:1, screw pitch of 6 mm, 201 parameter = 5000000, set the drive parameters corresponding to the coordinate axis station number; Setting Drive parameter 1/2/3/4/5/6 corresponds to: X/y/z/A/b/C, 6, MDI run M500 instructions, the small circle before the axis should become“Green”,

that read absolute encoder data correct; Set the soft limit values of the system drive parameters coordinates 9. If an absolute value is required system machine tool coordinates zero: Enter the machine tool coordinates values plus or minus the value multiplied by 1000, and then write the corresponding integrated parameters 205(x axis) , 206(y axis) , 207(z Axis) , 208(a axis) , 209(b Axis) , 210(c axis) .

M600 spindle returns to the zero position of the C-axis encoder as a positioning axis instruction

1、 Define the code, parameters, and wiring first:

1, M03, M04, and M05 are the speed control modes of the spindle switch.

2, M75 is the output port for pulse mode.

3， M600 is a user programming instruction for switching pulse modes.

4. Define the C-axis as the main axis pulse axis, so that the C-axis is used for positioning.

5. The servo spindle encoder signal must be connected to the CN9 interface of the system, and the welding wire signal is PA+, PA-,PB+,PB-,PZ+,PZ-

2、 The operation of the pulse mode encoder zero position signal is as follows: after programming M600, the action is as follows: close M03 or M04 (actually interlocked with M75), check the speed of the spindle encoder, until the speed is zero, then output M75, and automatically send out the C-axis pulse to make the spindle motor rotate. At this time, the screen below shows: the zero position signal position of the spindle servo is being returned (the speed of the motor is determined by the comprehensive parameter 38), until the zero position signal of the spindle encoder is detected, immediately stop sending the pulse, and then reset the absolute coordinate of the workpiece on the C-axis of the current tool position number to zero. The zero position action of the main axis encoder is completed. If any of the above actions are not completed, the screen below displays "hold", and the cursor keeps waiting for M75. On 600, the program cannot run downwards.

3、 The speed mode operation is as follows: After programming into M03, M04, or M05, first turn off the M75 signal (which is actually interlocked with M75) and then output M03, M04,M05。

Explanation of M601 spindle accurate stop positioning action

1、 Define the code, parameters, and wiring first:

1, M03, M04, and M05 are the speed control modes of the spindle switch,

2, M75 is the output port for pulse mode of the servo spindle driver

3， M61 is the positioning signal output port for the servo spindle driver,

4, M6133 (M28) is the input signal for positioning completion,

5, M601 is a user programming instruction that switches between pulse mode and positioning mode. This function does not define any axis names.

2、 The pulse mode operation is as follows: after programming M601, the action is as follows: close M03 or M04 (actually interlocked with M75), output M75 and M61, and then detect the positioning completion signal M6133 (M28). At this time, the screen displays: the automatic quasi stop of the spindle servo is running, until the M6133 signal is detected and M61 is closed. The screen displays cancel, and the positioning action is completed. If any of the above actions are not completed, the cursor remains on M601, and the screen displays hold. The program cannot run downwards.

3、 The speed mode operation is as follows: After programming into M03, M04, or M05, first turn off the M75 signal (which is actually interlocked with M75) and then output M03, M04,M05。

NG98TT multi-channel system operation instructions

1、 NG98TT is a dual channel and three channel CNC system that is fully compatible with single channel NG98T and NG98Ta CNC systems. Each channel can control 1-6 linkage axes X, Z, Y, A, B, C, with a total of 12 axes controlled by the dual channel and 18 axes controlled by the three channels. Each channel is equivalent to an independent lathe CNC system, which has independent editing, manual, automatic, tool offset, parameters, status, and processing programs. During operation, Each channel is independent of each other and does not affect each other,
However, in some machine tools or when processing certain special parts, it is necessary to coordinate the operation of multiple channels. That is, when certain instructions of the channel must be run in another channel before they can run, this system has designed corresponding multi-channel communication control instructions.

2、 Channel 1 of the display interface is a white display interface, which is displayed in red text in the upper right corner of the display screen: Channel 1. Channel 2 of the display interface is a black display interface, which is displayed in red text in the upper right corner of the display screen: Channel 2. Channel 3 of the display interface is a gray display interface, which is displayed in red text in the upper right corner of the display screen: Channel 3, each channel is an independent operation interface that needs to display and operate the function of another channel, Press the switch button to switch to another channel display interface and operation.

3、 The input and output interfaces behind the NG98TT system are independent for each channel. The dual channel input power supply is+5V and+24V, and the three channel input power supply is+24V. The shared channel functional interfaces are: hand pulse CN11, USB port, power supply CN8, and external switch CN18

4、 Multi channel shared function description:

1. Display interface shared, press the switch button to switch to another channel display.

2. Keyboard sharing, press the switch key to switch to another channel for keyboard operation.

3. USB sharing, press the switch button to switch to another channel for USB operation.

4. Hand pulse sharing, note: After the hand pulse is turned on, the current display channel is valid. If you need to switch channels, you must first turn off the hand pulse of the current channel, and then press the switch button to operate another channel. If the hand pulse function of the current channel is not turned off, switching channels and then operating the hand pulse function will cause the hand pulse of other channels to be valid, causing the axis of other channels to move and collide with the knife.

5. External switch sharing function, emergency stop sharing channel function, three position pause switch outputting channel 1, channel 2, and channel 3 respectively, independent control operation, total start sharing channel. After pressing, all channel programs can be started. If a separate channel program needs to be run, press the start button on the keyboard to display the current channel and run the current channel program, while other channel programs will not run.

5、 Waiting instructions between channels: M1800

1. Explanation: Each channel of the user program must be programmed into M1800, and the channel program to M1800 must be executed first. Wait until the other channel user programs run to M1800, and then all channels will run the next program segment simultaneously

2. If there is no need for waiting function programs between channels, do not include them in M1800

Definition of Input Signal Switch and Output Signal Interface

一、 Input signal M16, switch output M10

Input signal M14, switch output M79

Input signal T01 switch output M75

Input signal T02 switch output M32

Input signal T03 switch output M61

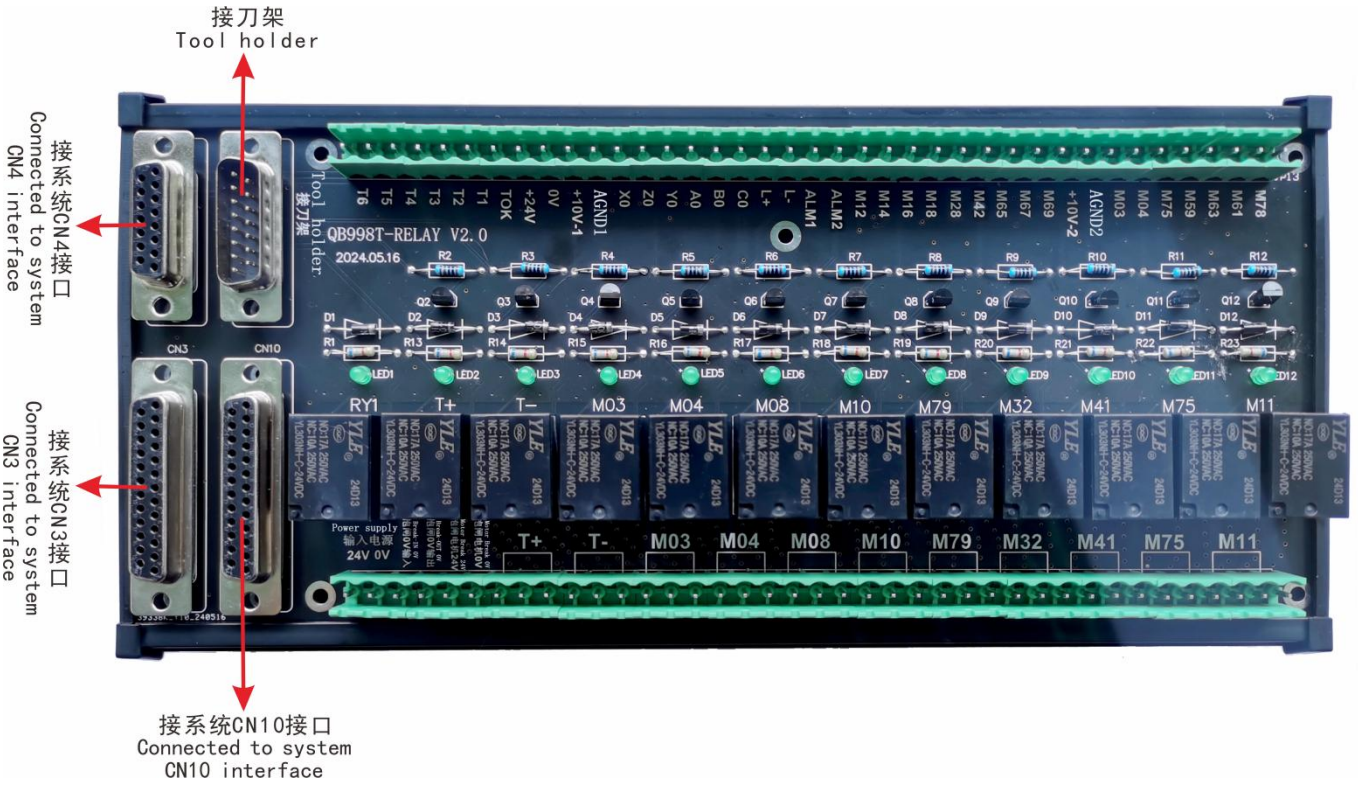
Input signal T04 switch output M63

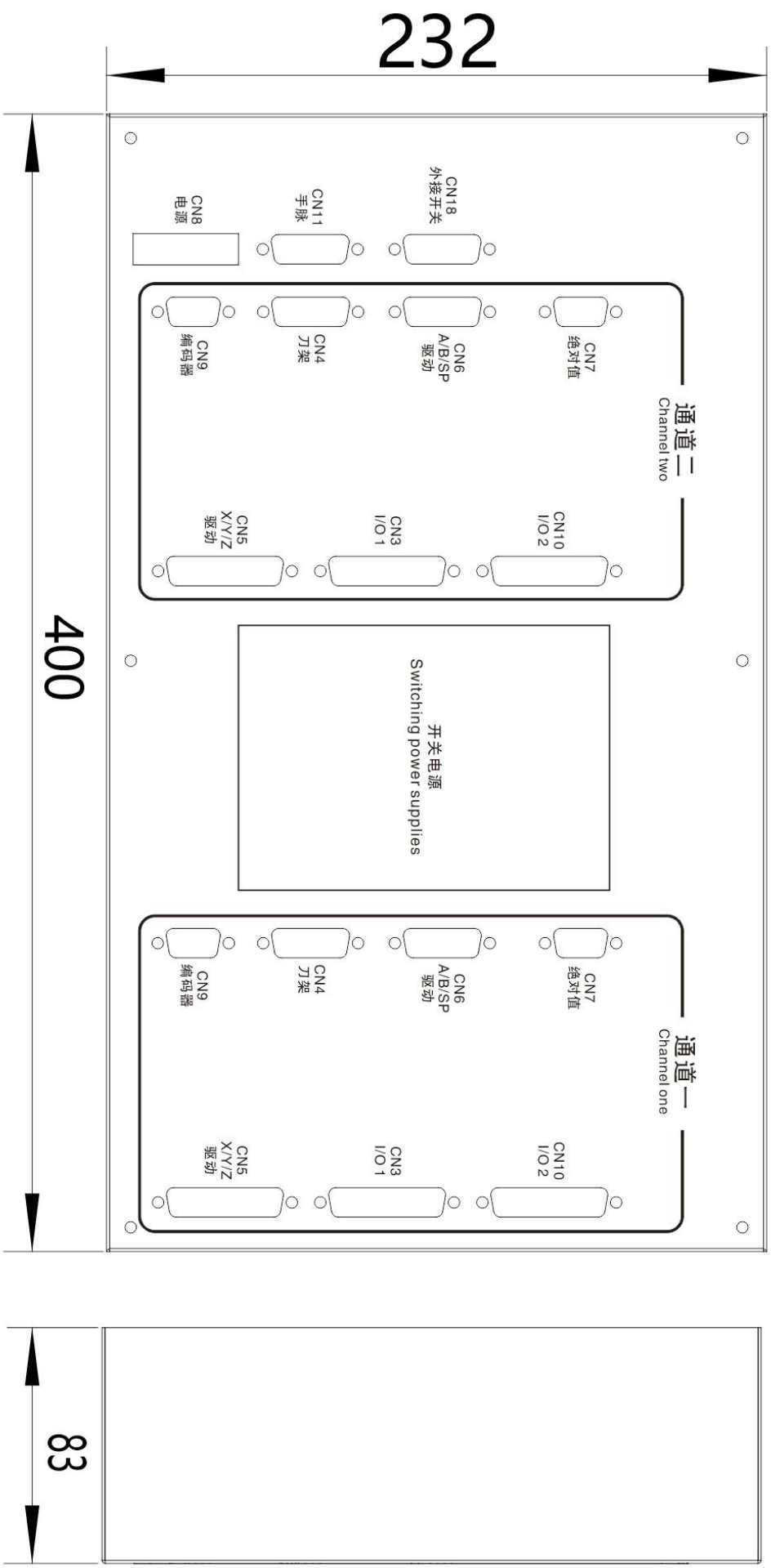
Input signal T05 switch output M59

Input signal T06 switch output M7116 (Y24)

Input signal T07 switch output M7117 (Y25)

Explanation: Are the external M10 and M79 of parameters 8 and 9 valid, and are M75, M32, M61, M63, M59, M7116, and M7117 valid for parameters 43 and 44.




232
400
83
998TT Rear view
CN8
电源
CN11
手脉
CN18
外接开关
CN9
编码器
CN4
刀架
CN6
A/B/SP
驱动
CN7
绝对值
CN5
X/Y/Z
驱动
CN3
I/O 1
CN10
I/O 2
通道二
Channel two
开关电源
Switching power supplies
通道一
Channel one