华兴数控
WASHING

Lathe numerical control system

USER MANUAL

Suitable for: 31XTA/32XTA/300T

99TA /99TB/99TY/99UZ

NANJING WASHING CNC Technology CO.,LTD

Content

Chapter One System Introduction

1.1 Main specifications

Pulse equivalent： X：0.001mm Z：0.001mm

Control/compounding axis number：2/2

Programming scope：±99999.999mm

Forward speed：60000mm/min（0.001mm equivalent）

Program capacity:：Electronic disk 640K, which can store 200 programs,10 parameters

Interpolation：Line, arc, metric system, English system straight/cone, multi-thread/single thread, tap

1.2 System resource

Display： 31XTA,32XTA,300T,99TA,99TB,99TY,99UZ system：7” LCD is adopted with 480×234 dot matrix

Electronic disk： 640k flash memory, which can store 200 programs and parameters

Input signal： 31XTA,32XTA,99TA,99TB,99TY system:33-way on-off and optical isolation
The zero signal of machine tool is accessed via the interrupt mode and adopts quick-response
300T,99UZ system:20-way on-off and optical isolation
Hand wheel interface: 1way, ×l, ×10 and ×100 multiply factor
99T system : :Hand wheel interface:2way, ×l, ×10 and ×100 multiply factor

Encoder interface： 1 way, four-multiple frequency processing

Output signal： 31XTA,32XTA,99TA,99TB,99TY system:17 ways on-off in total, Among them there are 13 ways relay power driving output and 4 ways relay contact output; driving signal (CP, CW) output in X ,Y and Z directions
300T,99UZ system: there are 15 ways relay power driving output and 2 ways relay contact output；And the X-Z triple-phase step motor interface is adopted
1 way 10-bit analogue output, output scope:0-10 V

Communication： RS232C asynchronous serial port, USB interface

Time： Processing time

1.3 Brief introduction of system major functions

Program management functions： Full-screen editing (ISO code), rename, delete, serial input/output，USB input/output and so on

Operating functions： Automatic, manual, pulse control, hand wheel, MDI, machine tool returning point, single segment, pause, coordinate and cutting tool memory and optional segment startup.

Parameters： Cutting tool parameter, clearance compensation, system parameter, digit parameter, thread pitch error compensation and so on.

Figure： Real-time track processing figure

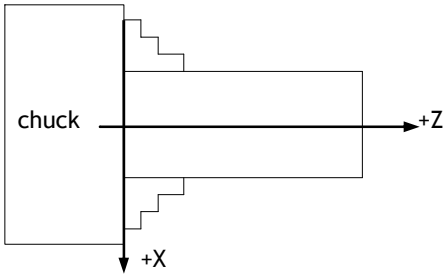
1.4 Coordinate system provision

When the elements are processed on the numerically controlled machine tool, the relative movement between cutting tool and elements must be in the specified coordinate system, and then the program can be processed according to provisions. For the convenience of describing machine tool

movement when programming, simplifying the program compilation methods to guaranty the exchangeability of recorded data, and the coordinate and movement direction of numerically controlled machine tool have been standardized. The Ministry of Engineering Industry promulgated the denomination standards of JB 3051-82 numerically controlled machine tool coordinate and movement direction in 1982, and the denomination principles and provisions are as follows

1.4.1 Movement principles which are relative to stationary work piece

This principle is for programming staves to determine machine tool operating process according to element drawings under condition that they don't know whether it's the cutting tool movement or work piece movement.



F1-1 Standard coordinate system

1.4.2 Standard coordinate system provision

The standard coordinate system is one rectangular coordinate system It's as Figure 1-1。

Various coordinate axes of this coordinate system parallel to primary guide tracks of machine tool

1.4.3 Provision on machine tool moving component direction

The movement positive direction of certain moving component of machine tool is the one to increase distance between cutting tool and work piece

1.4.3.1 Z coordinate movement

The Z coordinate movement is provided by spindle which transmits cutting power. In the standard coordinate system, the coordinate always paralleling to spindle is provided as Z coordinate.

1.4.3.2 X coordinate movement

The X coordinate is horizontal, which parallels to clamping surface of work piece. The X coordinate is the movement primary coordinate in the locating planes of cutting tool or work piece

On the lathe, the Z coordinate positive direction is that the big tool carriage moves to end bracket side along lathe bed (vertical), and X coordinate positive direction is that holder moves to handgrip direction

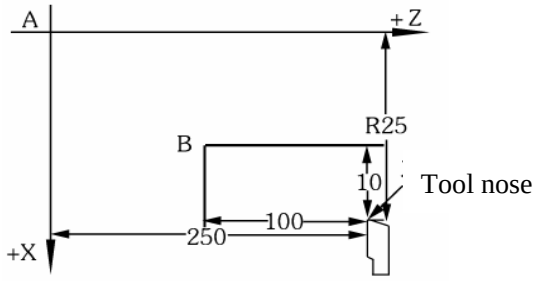
1.4.4 Machine tool reference point

The machine tool reference point is also called mechanical zero, which indicates X and Z directions to move to the approaching limit position along positive direction, and induces the determined position of reference point switch in this direction. Whether one machine tool has reference point returning function depends on whether machine tool manufacturer installs reference point switch (also called mechanical origin switch)

1.4.5 Work piece coordinate system

Select the fixed position on the machine tool as the origin, and the coordinate value corresponding to this origin describes the coordinate system of the work piece shape; moreover, the programming of general work piece is implemented based on the work piece coordinate system. Take the lathe for example, the origin of X-direction is the axial line of the work piece, and the origin of Z-direction selects the chuck end surface or the work piece end surface。

All the coordinate values are positions of tool nose relative to coordinate origin. If the Coordinate origin is different, even if the tool nose is at the same absolute position on the machine tool, its



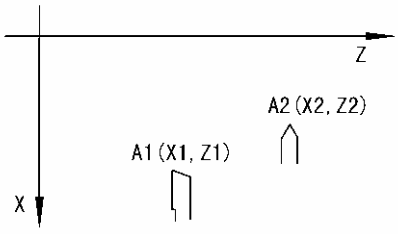
F1-2

coordinate value is also different. In order to guaranty uniqueness of tool nose coordinate in processing, the coordinate origin (also called zero) must be determined, while zero position is acquired by opposite calculating of tool nose position and coordinate value

For instance: If the tool nose coordinate is supposed to be (50, 250), then the 25 mm position along X negative direction is the X coordinate origin; the 250 mm position along Z negative direction is the Z coordinate origin (see A position in the right figure)

Note: On the lathe, the X direction coordinate (also called horizontal direction) is provided as diameter amount.

Now supposing that the tool nose position is unchanged, while the coordinate is (20,100), then zero is in the B position in the figure, which is the notion of floating zero. However, towards to one processing program, the processing can be implemented only after determining zero and optional change is prohibited (unless through instruction of coordinate transition)Once the floating zero is determined, the work piece coordinate system used in the actual processing will be composed. All tool nose movements in the program are based on this coordinate system for reference. Seeing G92 instruction to determine coordinate zero



F1-3

1.4.6 Cutting tool compensation principle when changing cutting tool

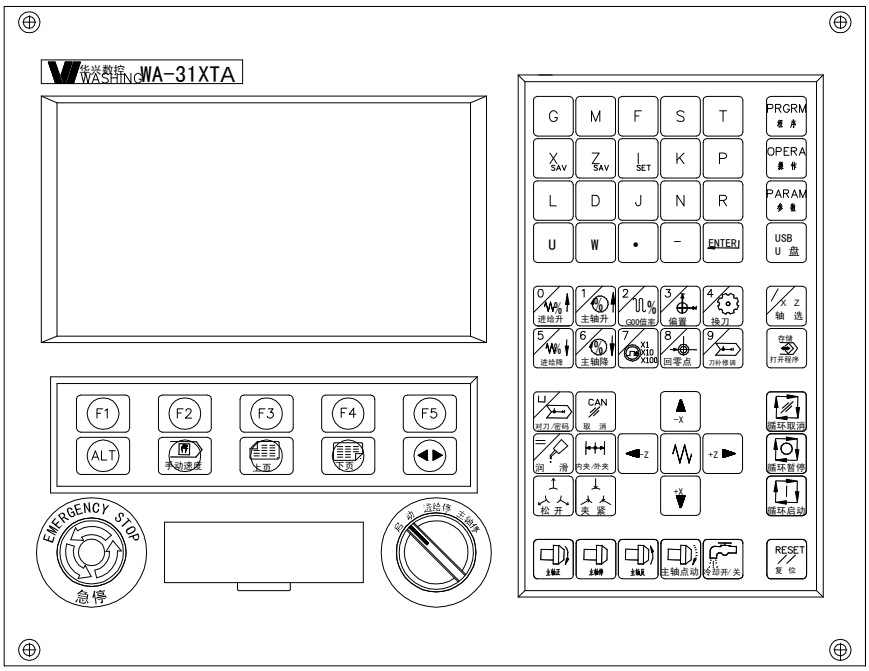
When the comparatively complicated work piece is processed, more cutting tools are frequently required. However, the processing program is compiled according to some one tool nose of cutting tool. After changing cutting tool, the offset must occur in X and Z directions of current tool nose relative to previous one. That's to say even if big and small tool carriages don't move, the tool nose position will change after changing cutting tool, and the effect of cutting tool compensation is used to compensate this change

For instance: If current cutting tool is T1, its tool nose position is A1; after changing for No. 2 cutting tool (T2), the tool nose of it is in the A2 position. The tool nose coordinate is changed from A1 (X1,Z1) to A2 (X2,Z2) after changing cutting tool, and the effect of cutting tool compensation is to convert the original coordinate (X1,Z1) of tool nose coordinate value to (X2,Z2). The relative difference in X and Z directions of A1 and A2 can be measured beforehand, and this difference is the cutting tool compensation value memorized by numerical control system. In practical applications, in order to simplify this course, the numerical control system doesn't measure difference of each other among cutting tools, but adopts simpler methods to memorize cutting tool compensation value. That's the method of memorizing coordinate value for determination.

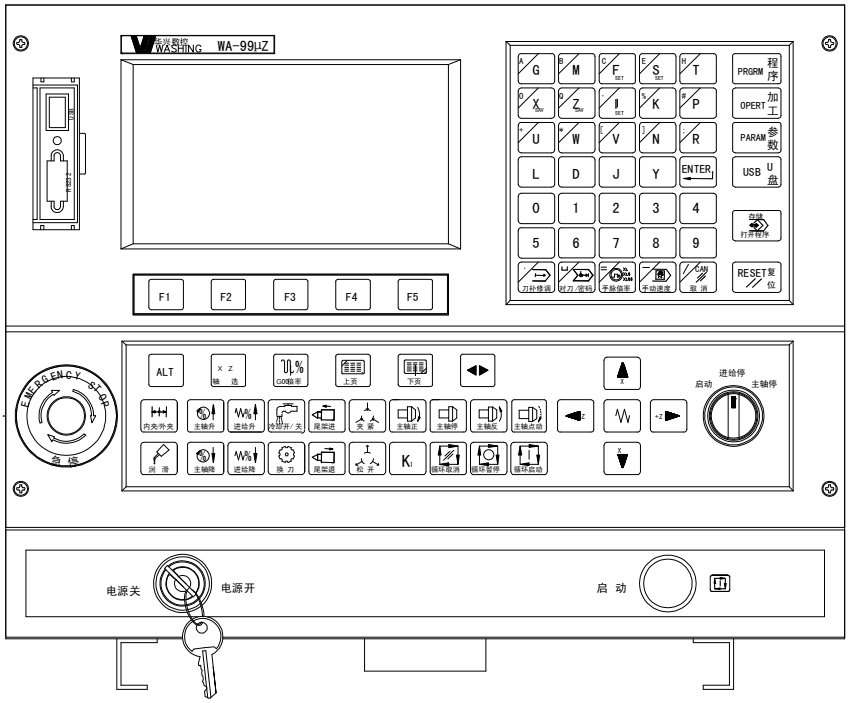
For instance: Contacting certain fixed point (core rod or specimen) one by one along X and Z directions for tool nose of each cutting tool, and this fixed point contacted by tool nose is regarded as standard. Owing to difference of various cutting tool lengths, the displayed coordinate point is also different contacting to fixed point. The numerical control system respectively memorizes coordinate values contacted by various cutting tool. These different coordinate values of each other actually include the length difference information between the two cutting tools. Many methods can be utilized to generate cutting tool compensation value, and 31DT adopts the method that inputting work piece size after one cut, which is equivalent to regard the presetting cutter as benchmark. It can calculate cutting tool compensation values of encircle, internal hole and so on, moreover eliminate errors brought about by process system elastic deformation

1.5 Operating keyboard of numerical control system

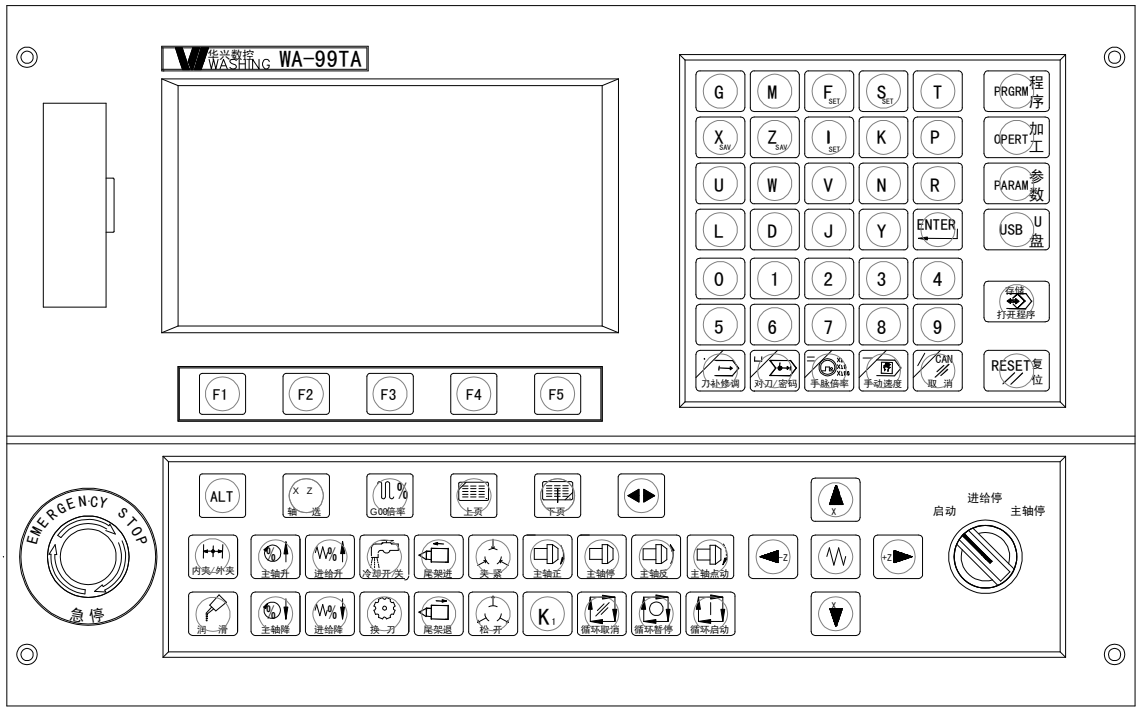
The whole operating functions offered by numerical control system can be implemented by keyboard operation。The front panel is composed of 6 Inch LCD, address function keyboard region, numeric keyboard region and manual operation keyboard region. The operating panel of system is as the figure below:



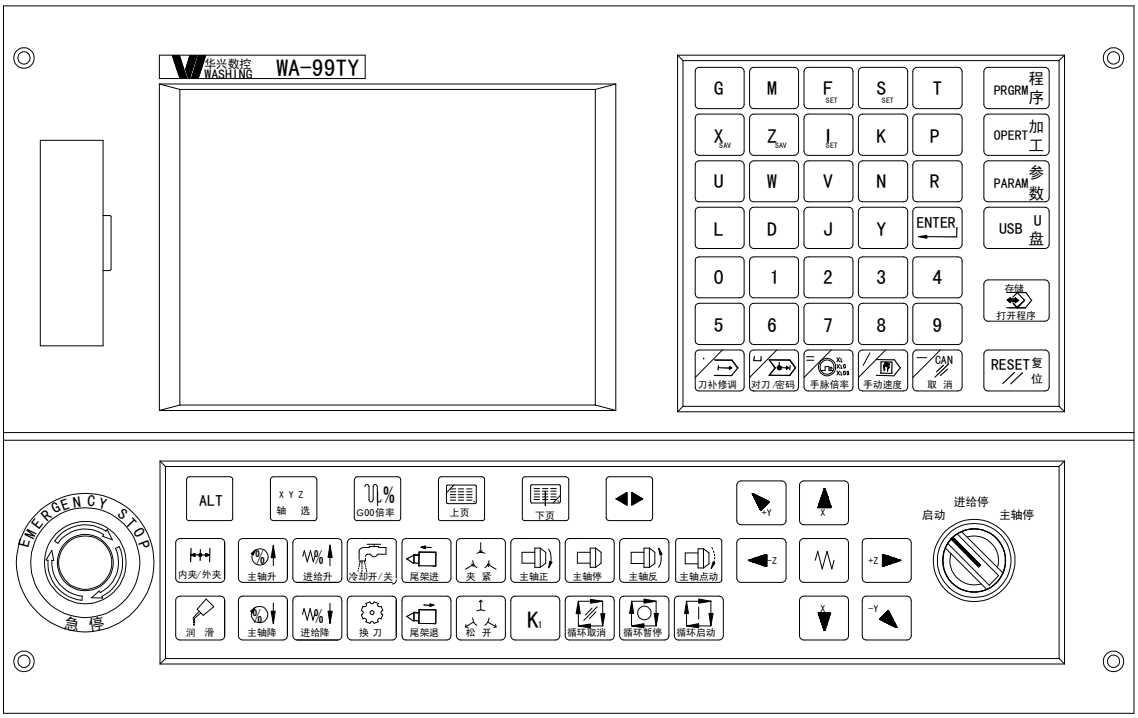
F1-4（a）31XTA/32XTAsystem front panel outside drawing



F1-4（b）99UZ/300T system front panel outside drawing



F1-4（c）99TA/99TB system front panel outside drawing



F1-4（d）99TY system front panel outside drawing

numerical control system has four major functions

PRGRM-Program: Various management, program input and output operation related with program.

OPERT——Process: All movements and strong signal control related with machine tool.

PARAM——Parameter: The control parameters are set according to various machine tools

USB------USB interface management function

1.5.1 Primary function key

The primary function key region includes 5 keys of program, operate, parameter, monitor, and reset,

which are used for the 5 primary functions of the system. Their detailed functions and meanings are shown as table 1-1.

Table 1-1 Description to the key-presses in the primary function keyboard region

Key-press	Symbol	Description
PRGRM 程序	【Program】	The user processing program management which is used to manage the processing program compiled by the user, and under this primary function there are sub-functions related with program management
OPERT 加工	【Process】	It's the machine tool operating, towards various operating functions, they can be implemented in the sub-functions under this function.
PARAM 参数	【Parameter】	It's the parameter setting which is used to set various parameters related with machine tool or numerical control system
USB U盘	【U disk】	USB interface is used to administer switch program and parameter between the U disk and the system.
RESET 复位	【Reset】	It's the soft reset, which is used for canceling the operation being implemented, and shutting down the cooling liquids, spindles, and the cutting tool output signals.

1.5.2 Soft definition key F1~ F5

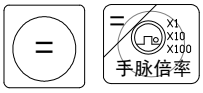
There are five keys marking F1~F5, the function of the key will change with the primary function selected by current user, which is primarily used to select sub-function belonged to the primary function under it. The meaning of F key is corresponding to the Chinese character in the lower part of current screen. When exceeding 5 keys, pressing "◄►" key to switch to soft function menu of the next page.

1.5.3 Edit character key

It's primarily used to input ISO code and parameter values of various coordinates of processing program

Table 1-2 Description to the key-presses in the editing keyboard region (edit function key)

Key-press	Symbol	Description
G … P	【G】…【P】	Letter key. With G M F S T X Z I K P L D J N U W R
0 … 9	【0】…【9】	Number key. With 0 1 2 3 4 5 6 7 8 9
CAN 取消	【 】	Space key, it'll be used as Space for program editing
− 对刀	【－】	Minus key, the next key is used in 97T/98T, and it'll be used as minus for editing and inputting
. 刀补修调	【.】	Decimal point key, the next key is used in 97T/98T, and it'll be used as decimal point for editing and inputting

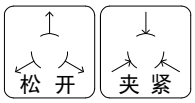
	【=】	Equal sign key, the next key is used in 97T/98T, it'll be used as equal sign for editing and inputting

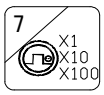
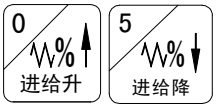
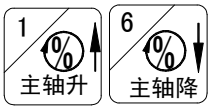
1.5.4 Description to the key-presses in the editing keyboard region (multiplex function keys)

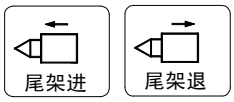
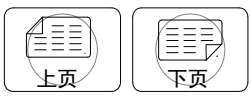
Key-press	Symbol	Description
	【Z－】	Manual feed in the Z negative direction.
	【Z＋】	Manual feed in the Z positive direction.
	【X－】	Manual feed in the X negative direction.
	【X＋】	Manual feed in the X positive direction.
	【Fset】	F Set, is used for manually setting up the feed speed.
	【Iset】	Set the step quantity I
	【Sset】	S Set, is used for manually setting up the rotation speed of the spindle.
	【Xsav】	X Save, is used for saving the numeral value of the X axis in the current programming coordinate system, as the measuring cutting tool compensation value parameter.
	【Zsav】	Z Save, is used for saving the numeral value of the Z axis in the current programming coordinate system, as the measuring cutting tool compensation value parameter.

1.5.5 Other

Table **1-3 other**

Key-press	Symbol	Description
	【Spindle clamp】 【Spindle loose】	Spindle clamp/loose
	【Clamp inward or outward】	Clamp inward or outward
	【Manual speed】	Manual speed
	【Axis selection】	It is the selection of the machine tool movement axis of feed control under the manual wheel mode.

	【Cycle start】	Cycle start. It starts the processing and implementation of the current programs.
	【Cycle pause】	Cycle pause
	【Cycle cancel】	Cycle cancel. It terminates the automatic cycle processing operation of the current program
	【Manual fast feed】	Manual fast feed option key. Under the non-modality condition, pressing it with any one key in the manual feed keys, the machine tool moves in high speed, and the moving speed is set up by the No. 10 system parameter
	【Hand wheel pulse】	Set hand wheel pulse multiply factor. The setting range: **×1**、**×10**、**×100**
	【Coordinate bias】	Coordinate bias
	【Feed increase】 【Feed decrease】	dynamically adjusting the feed multiply factor
	【Spindle increase】 【Spindle decrease】	dynamically adjusting the spindle multiply factor (only valid for main motor speed control by frequency variation).
	【Cutting tool compensation setup】	This key is the shortcut one, under status of manual, automatically and hand wheel operating; the cutting tool compensation parameter interface can be fast accessed by pressing this key.
	【SAVE】 【OPEN】	When the system is in the primary function interfaces of program management or parameter management, press this key to indicate the saving of the files such as the current programs, system parameters, cutting tool parameters, machine tool parameters etc. When the system is in the status of processing, press this key to indicate the calling of the user programs, then the current interface ejects the program name list and the entry box of the program names, then input the program name in the entry box of program names and press ENTER, to indicate the calling of this program.

	【Cutting tool-change】	Single-step tool-change: The tool will be changed to the next in turn by the system for each pressing
		Spindle clockwise rotation
		Spindle counterclockwise rotation
		Shutting down the spindle, and stopping rotation
		Spindle pulse control. The spindle rotates clockwise when this key is pressed, and the stops rotating when this key is released
	【Tailstock forward】【Tailstock backward】	
	【Cutting tool compensation】	In operation and processing interface, quickly enter cutting tool compensation modification
	【Coolant on/off】	Coolant on/off
	【Lubrication】	
	【Return to presetting cutter point】	Return to presetting cutter point: It's equivalent to G76, and at the beginning off the work piece, using 18# and 19# to set one coordinate value as the processing start point, after processing completes every time, the holder return to this point, which can implement measurement and clamp work piece again and so on
	【CAN】	Clear wrong messages in the alarming bar，cancel input
	【the previous page】【the next page】	Page turning key. Showing the content of the previous page or the next page for the interfaces such as the program name list, program content, parameters, etc.
	【ALT】	When machine is switched on, press this key to enter upgrade interface. The display mode can be switched in the operation interface.

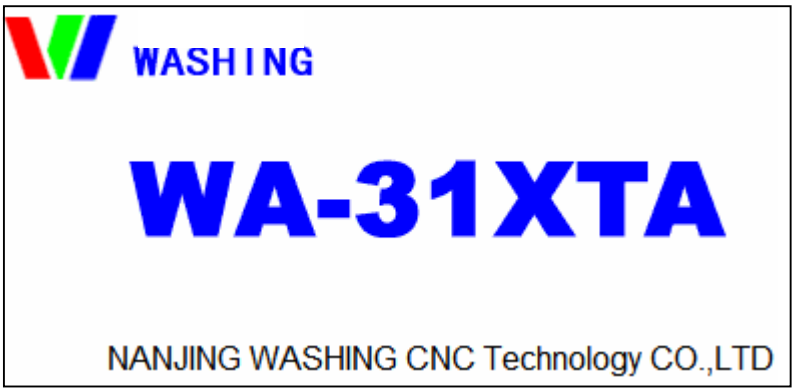
Note: Some keys in the editing keys are the function multiplex keys, which are used multiply by the system according to the current status.

1.6 Starting up

Before the first starting up, the system appearance should be examined to see whether there is obviously abnormal condition, whether the power connection is not right, whether the joint to the switch power comes off, and it can be powered on after confirmation. The system power source is the three core power plug, which uses single-phase 220V / 50HZ A.C. with grounding Connect the earth wire, connect the grounding copper bar of the machine tool strong power cabinet

1.6.1 Starting up, image and design

When the system starts up, it displays the image of starting up



F1-5 System starting up interface

The system start-up interface can be customized, and the user can customize a 256-color picture with 480*234 matrix

a) Download it to the system from U disk, and the specific process is as follows：

1. When the machine is switched on, press ALT key until the upgrade interface appears

2. Save the picture in the root directory of U disk, and modify the file name according to the prompt of upgrade interface, and then insert the U disk.

3. Press 【F1】(U disk) and then press 【F4】(start-up interface), the system is upgraded.

b) Download it to the system from Serial commutation port, and the specific process is as follows：

1. When the machine is switched on, press ALT key until the upgrade interface appears

2. Press 【F2】(and then press 【F4】(start-up interface), the system is upgraded.

1.6.2 Selection of primary function

After starting up, only the primary function selection is valid, if user wants to use some one specific function, it can be implemented only after accessing relevant primary function. Accordingly, the user must press the selection key of primary function which includes the specific function (sub function) first of all, and selects desirable function after accessing corresponding primary function status. The operating interface of this system adopts note mode as far as possible, which makes user learn about whether the current operation is offered by the system. Simultaneously, the principle of key-press operation is: Implementing user's desirable functions with fewest key-press numbers.

Generally, in order to implement some one specific function operation, the user simply presses the key three times

1. Pressing the primary function key, the system will quit original status and access new primary function status.

2. Pressing sub-function selection key (it's one of F1～F5 in general), and setting sub function valid.

3. Under sub function status, pressing specific operating key to implement specific function. For instance, the current system is in the editing mode and is editing processing program, the user hopes to access operating status and moves X and Z axes to proper position, and the operation is as the following sequence:

(1). Pressing OPERT (Process) primary function key, the system will automatically save the edited processing program, then quit "PRGRM" primary function and access "OPERT" primary function, the main image of "OPERT" is displayed in the screen.

(2). Pressing coordinate moving keys-、Z+►、X-▲、X+▼ to move coordinate axis。

This is one general operation, 2 to 4 times of operation are probably required towards other conditions。

1.6.3 Selection of sub-function

There are 5 keys marking F1～F5 in the lower part of screen, the roles of these five keys are to select various sub-functions under some one primary function. Owing to that primary function has three kinds while various sub-functions under primary function are also different, accordingly the roles of F1~F5 change at any moment, and towards to current specific definition of F1~F5, the notes are displayed in the screen. Accordingly, F1~F5 are also called soft definition keys or F function keys. Towards to the undefined F key under current primary function, there are no corresponding notes in the screen in general, and the system makes no response with pressing this key. This system will probably define it when the software version upgrades. When F1~F5 are not enough, the key "◄►" can be pressed to switch to the next page of F function key.

Chapter Two Programming

2.1 Block format

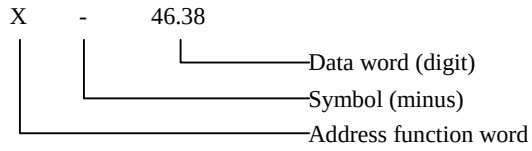
The block format means the written rules of block, it includes the function which will be implemented by numerically controlled machine tool and parameters that are required implementing this function. One part processing program is composed of several blocks, and each block is composed of various functions; General function words of lathe numerical control system are as follows：

Table 2-1: Table of system function words

Function	Address	Scope	Meaning
Program number	P、N	01～99	Specify program number and sub-program number
Sequence segment number	N	0～99999	Program segment number
Preparatory function	G	00～99	Instruction movement mode
Coordinate letter	X、Z、I、K、R、L、J、D、	±0.001～±99999.999	Movement instruction coordinate, circle center coordinate, thread pitch, radius and cycle number
Feed speed	F	1～15000mm/min	Feed speed instruction
Spindle function	S	0～5000RPM	Instruction of spindle rotating speed
Cutting tool function	T	1～8	Cutting tool instruction
Auxiliary function	M	0～99	Auxiliary instruction

The numerical control system doesn't require each block having these instructions above, however in every block; the instruction should be arrived according to certain format. Each function word probably has various definitions in different block definitions, referring to specific instructions.

Washing numerical control system adopts the variable block format, which means that the block length changes with word number and word length. One block is composed of one or more program words. Generally, the program word is composed of address word and post-address word number and symbol, for instance:



This program word format is composed of address function word as the header following a string of digits, and several words compose one block. The words written in the previous block and unchanged in this block are also valid, which can be no longer rewritten. In the dimension words, it can only write valid digit and not specify that every word should write the full number.

For instance：N0420 G03 X70 Z-40 I0 K-20 F100

In the previous segment of program:

N、G、X、Z、I、K、F are all address function words

N	Block number
G03	Preparative function, which can be written as G3
X Z I K	Coordinate address
F	Amount of feed

"-" Signifying symbols

03 70 -40 0 -20 100 are data words

In the block, the English letters signifying address function can be divided into dimension word address and non-dimension word. The dimension word address is signified with the following letters X、Z、I、K、R、J、D, and non-dimension word address is signified with the following letters: N、S、T、G、F、M、P、L. All dimensions are signified with diameter or diameter difference in X direction. For instance, X50 signifies that tool nose moves to Φ50 position, and I10 signifies that the diameter difference of circle center relative to arc start point is ΔΦ10.

One complete program is composed of program name , block number and corresponding symbols, Refer to the following instances for discriminating different programs and program contents in the program directory:

```
N0010   G92   X50   Z100
N0020   S1200   M03
N0030   G01   X40   F300
N0040   Z90
N0050   G02   X30   Z85   I0   K-5
N0060   G01   Z60
N0070   G02   X40   Z55   I10   K0
N0080   G01   X51
N0090   G0    X50   Z100
N0010   M02
```

In general condition, one block is one process step of element processing, numerical control program is one block statement sequence which is stored in the memorizer. When the elements are processed, these statements are totally read from memorizer and explained into executable data format and then executed.

The block number is used to identify every block composing program; it's composed of N with following 0000-9999. The block number must be written at the beginning of every segment, which can be generated by segment number automatic generator. (See program edit function) In one program, the block number can adopt optional value in 0000-9999, however in principle, various block numbers should be arrayed from small to big according to its precedence in the program. For the convenience of inserting new block in the required position, it's recommended not to use continual serial no. to the block when programming, and if the programming is implemented on the CNC panel, it's suggested that the block be numbered with interval of 10. In this way, it's easy to assign different block numbers when inserted into the program. (See parameter P27#).

2.1.1 Macro-variable

The parameters (P0-P9) can be used to substitute digit in the block. First the assignment statement in the program can be used for parameter (P0-P9) assignment, and in the subsequent programs, the assigned parameter can be used to substitute this numerical value. When the program is automatically executed, the parameter is changed to the lately assignment number of this variable. If this parameter is assigned again in the program, then the new value is only valid to the changed quotation, and the previous quotation still remains the original value.

For instance:

```
N0010   P2=1     P5=55     P7=200
N0020   G92      XP5       Z100      FP7
N0030   F2=40    P5=160
N0040   XP2      ZP5
N0050   M02
```

When the program is executed, it's identical with：

```
N0020   G1   X55   Z100   F200
N0040   X40  Z160
N0050   M02
```

2.2 Preparative function (G function)

The preparative function is programmed with G following two digits, G function is also called preparation function command, which is used to define geometry and CNC operating status of track. The functions of any numerical control device all include fundamental functions and optional functions. The fundamental functions are requisite for system, and the optional functions are for user to select according to machine tool features and applications. It's recommended to program after understanding the machine specifications first. The machine tool can configure control function according to numerical control system, namely the machine tool may not implement all functions of numerical control system.

The whole G functions of system are as follows:

Type	G Code	Function
Modality	G00	Fast locating
Modality	G01	Line interpolation
Modality	G02	Interpolation of clockwise arc
Modality	G03	Inverse circle interpolation
	G04	Time delay
	G09	Servo accurate positioning pause
	G10	Cancel various image processing cycles
	G11	Image processing cycle of plane figure along X axis (suitable for milling processing)
	G12	Image processing cycle of plane figure along Y axis (suitable for milling processing)
	G13	Image processing cycle of plane figure along zero point (suitable for milling processing)
	G17	Choose cutter compensation plane as XOY (suitable for milling processing)
	G18	Choose cutter compensation plane as ZOX (lathe control automatically chooses ZOX plane cutter compensation)
	G19	Choose cutter compensation plane as YOZ (suitable for milling processing)
	G20	Independent sub-program call
	G22	Independent sub-program definition
	G24	Return call program after sub-program ends
	G25	Jump processing
	G26	Transfer processing (sub-program call in the program)
	G27	Endless cycle
	G28	Metric system varying pitch single-tool thread
	G29	English system varying pitch single-tool thread
Modality	G30	Magnification and minification cancel
Modality	G31	Magnification or minification
	G33	Metric system single cutter thread processing cycle

	G34	English system single cutter thread processing cycle
	G40	Cancel cutting tool (tool nose) radius compensation
	G41	Cutting tool (tool nose) radius left compensation
	G42	Cutting tool (tool nose) radius right compensation
	G43	Cutting tool length compensation (suitable milling processing)
	G44	Cancel cutting tool length compensation (suitable for milling processing)
	G50	Processing program temporary modification system parameter
	G54	Cancel zero bias, recover the workpiece coordinate returning to mechanical origin in power-on
	G55	Absolute value zero point coordinate bias
	G56	Increment value zero point coordinate bias
	G57	Coordinate bias of current coordinate point
	G61	Fast corner clearing of subsequent block
	G62	Fast corner clearing of current segment
	G64	Cancel G61 corner clearing function
	G71	Internal (external) diameter cutting compound cycle
	G72	End surface cutting compound cycle
	G73	Sealed contour compound cycle
	G74	Return to reference point (mechanical origin)
	G75	Return processing start point based on machine tool coordinate
	G76	Return processing start point based on workpiece coordinate
	G78	Fine boring processing cycle (suitable to milling processing)
	G79	End surface screw thread metric system
	G80	End surface screw thread English system
	G81	Ex-circle (inner circle) fixed cycle
	G82	End surface fixed cycle
	G83	Deep hole processing cycle
	G84	Metric rigidity threading cycle
	G85	English system rigidity threading cycle
	G86	Metric thread cycle
	G87	English system thread cycle
	G88	Fine boring processing cycle Ⅰ (suitable to milling processing)
	G89	Fine boring processing cycle Ⅱ (suitable to milling processing)
Modality	G90	Absolute value mode programming
Modality	G91	Programming with incremental mode
	G92	Modify coordinate zero point position of workpiece coordinate system (change workpiece coordinate value of tool nose)
	G96	Effective for constant linear speed cutting
	G97	Cancel constant linear speed cutting
	G98	Cancel feed of every rotation
	G99	Set feed of every rotation

NOTE: One part of the G function above is suitable for lathe, and one part is suitable for milling machine and one part is suitable for both; the detailed description of G function is final, in this manual, the functions t the processing of milling won't be described。

2.2.1 G00——Fast locating

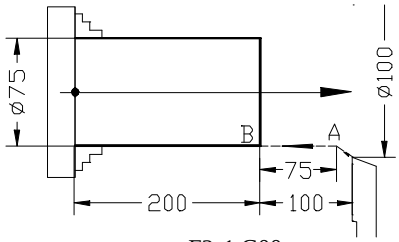
Format：G00 X _Z_

Description: (1) X & Z axes respectively moves at the rate which is defined by 06# and 08# parameter, and when certain axle stops after completing programming value, while other axes move on.

(2) The non-movement coordinate requires no programming.

(3) The coordinate value of target point can use absolute value, and can also use increased value. Most 6 digits are allowed before decimal, and after decimal most 3 digits are allowed, the positive number can omit "+" (this rule is suitable for all coordinate programming).

(4) When G00 is programmed, it's also written as G0.



F2-1 G00

Example：The right figure program is as follows:

Absolute value mode programming：G00 X75 Z200

Incremental mode programming：G91 G00 X-25 Z-100

First, X and Z simultaneously move fast to A point with 25, then Z moves fast to B point with 75.

Note: For the third axis control, Y axis can be directly programmed as follows:

G00 X50 Y120 Z32

Interim description between program:

The interim includes two modes between two segments of processing traces: arc switchover and corner clearing (sharp corner) interim, refer to 4.3.1.4 and description of G61, G62 and G64

2.2.2 G01—Line interpolation

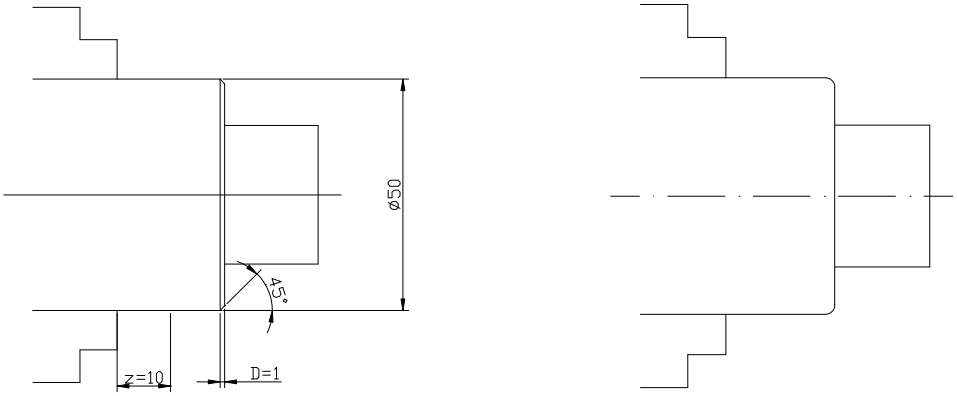
Format1)： G01 X__Z__F__

For the chamfer in mechanical processing, G01 can be provided with chamfer function

Format2)：G01 X 50 D 1 [R 1] F 100

Z 10

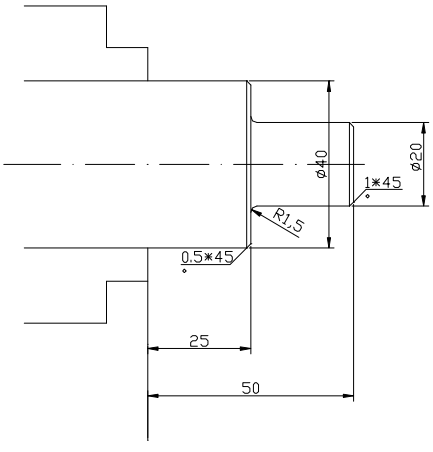
D belongs to the incline 45° chamfer and D is the single side distance as F2-2



a)Straight line chamfer b) Arc chamfer

F2-2

R is the arc chamfer, D or R is maximally 10mm, and X and Z only support processing along end surface or ex-circle (namely single side processing), the processing along slanting line isn't supported, else 05# error will occur.



The chamfer program can occur continually：

N0010 G00 X0 Z50

N0020 G01 X20 D1 F100

N0030 Z25 R1

N0040 X40 D1

N0050 Z5

N0060 G0 X50 Z100

2.2.3 G02——Interpolation of clockwise arc

Format：G02 X_Z_I_K_F_

G02 X_Z_R_F_

Description: (1) When X and Z are at G90, the end coordinate of arc is the absolute coordinate value which is relative to programming zero. When it's at G91, the arc end is the increment value of relative arc start point. Towards to G90, G91, I and K, they are all the coordinate values of center relative arc start point, and I is the diameter amount in X direction value, K is Z direction. The circle center coordinate can't be omitted at the circular interpolation, unless it's programmed with R (arc radius).

(2) When G02 instruction is programmed, it can directly program over the quadrant circle and full circle and so on (R programming can't be used for full circle).

Note: When arc is processed to the top, X or Z axis will probably change direction, and system will automatically perform clearance compensation; if clearance compensation isn't input in parameter zone or the difference between clearance compensation in parameter zone and practical reverse clearance of machine tool is big, obvious chipping mark will generated on workpiece.

(3) The full circle can't be programmed with R.

(4) R is the R arc's radius of work piece single side which is the number with symbol, "+" indicates that the arc angle is less than or equal to 180 degree; "-" indicates that the arc angle is greater than 180°.

(5) G02 can also be written as G2.

Example：The AB segment arc program of processing figure 2-3 is as follows:

Absolute value mode:

G90 G02 X60 Z30 I20 K0 F150（circle center coordinate programming）

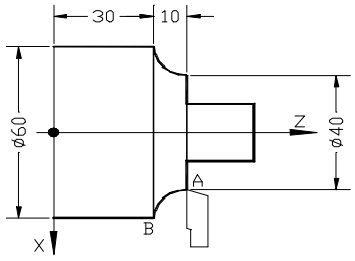
G90 G02 X60 Z30 R10 F150（radius R programming）

Increment mode:

G91 G02 X20 Z-10 I20 K0 F150(circle center coordinate programming）

G91 G02 X20 Z-10 R10 F150（radius R programming）

G02 U20 W-10 R10 F150



F2-3 G03

2.2.4 G03——Inverse arc interpolation

Format：G03 X_Z_I_K_F_

G03 X_Z_R_F_

Description: When programming with G03 instruction, except that the arc rotating direction is opposite, the rest are same as the G02 instruction.

Example: The program of Figure 2-4 is as follows:

Absolute value mode:

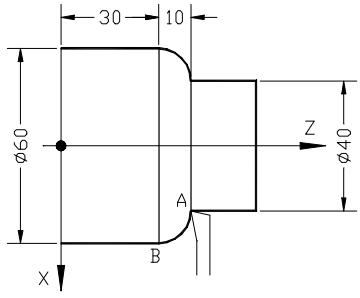
G90 G03 X60 Z30 I0 K-10 F100 (circle center coordinate programming)

G90 G03 X60 Z30 R10 Fl00 (radius R programming

Incremental mode:

G91 G03 X20 Z-10 I0 K-10 F100 (circle center coordinate programming)

G91 G03 X20 Z-10 R10 F100 (radius R programming)



F2-4 G04

G03 U20 Z-10 R10 F100

2.2.5 G04——Suspension

Format: G04 K×××.××

Description: (1) After the program delays post-K programming value (second), continue to operate, and the scope of time delay is from 0.01 second to 65.53 seconds.。

2.2.6 G09 ——Angle-specified stop of feeding

Format: G09

Description: G09 is used to detect whether servo-motor has moved to appointed position, When servo following error is less than given value, servo will send one angle-specified stop signal (XPSN、YPSN and ZPSN) to CNC system. After the system completes certain segment program, if this segment has G09, then CNC will check whether various axes have PSN signal input within a period of time, and this time is set by 89# parameter. If there is no signal when set time is exceeded, CNC will give 54# alarm to continue operation.

2.2.7 G20——Sub-program call

Format：G20 N××. ×××

Description: (1)The first 2-digit after N (to position before decimal) is the program name of sub-program which will be called and 2-digit is allowed. The 3-digit after decimal signifies the cycle number of this call that can be from 1 to 255.

(2) The parameter（P0～P9）in the sub-program must be assigned definite numerical value by P when it's called by G20.

(3) This segment of program mustn't appear contents outside of the descriptions above.

(4) Various sub-programs can repeat embedding call for 10 times, but calling itself is prohibited.

2.2.8 G22——Sub-program definition

Format：G22 N ××

Description: (1) The sub-program name is initiated with N, and two digits after N is the sub-program name.

(2). G22 Nxx mustn't be in the same segment with other instructions.

(3). G22 and G24 appear in pair, which forms one complete sub-program body.

(4) The parameter data in the sub-program has two kinds of formats:

a) Constant format, the data is constant set by programming, namely 0~9.

b) Parameter format, the numeric section such as function number, parameter and so on in the program can be signified with variable, while the specific value of variable is imported by P=×× definition in the main program of calling sub-program. This system can process 10 variable parameters: P0 P1 ···P9.

(5) Sub-program and transition processing (G25 and G26) can implement compound nesting for most 10 times.

(6) When it's necessary for parameter to define variables, P0＝××, P1＝×× and so on can be used to assign definite numerical values to P0#-P9#. No matter whether P parameter appears in the main program or sub-program, this parameter will be superseded with the lately assignment.

2.2.9 G24——Completion and returning of sub-program

Format：G24

Description: (1) G24 indicates completion of program and returns to the next segment of program calling this sub-program.

(2) G24 and G22 appear in pair.

(3) The segment of G24 disallows other instructions to appear.

Example: The parameter transmission process in the subprogram call will be described through the

following example, please apply it.

Main program P01

N0010 S1000 M03

N0020 P7=200 P8=50 P9=02

N0030 G20 N05

N0040 M02

Sub-program N05

N0010 G22 N05

N0020 G92 X50 Z100

N0030 G01 X40 FP7

N0040 Z97

N0050 GP9 Z92 X50 I10 K0 FP8

N0060 G01 Z-25 FP7

N0070 G00 X60

N0090 Z100

N0100 G24

Note: (1) If P parameter is not defined when subprogram is called, then the value of P parameter in the subprogram is indefinite.

(2) The parameter can also be used in the main program.

2.2.10 G25——Jump processing

Format：G25 N××××. ××××. ×××

Description: (1) The cycle body which is defined by this format is the defined block (including these two segments) between two block numbers following N, and the digit defines the call number of this block, from 1 to 255, and 1 will be considered without compiling.

(2) The next segment processing program after G25 instruction completes is the one of jump processing block.

(3) Other instructions are prohibited in the G25 block.

Example：

N0010 G92 X50 Z100

N0020 G25 N0040.0060.02

N0030 G00 X10 Z20

N0040 G01 X40 Z80 F300

N0050 Z60

N0060 G00 X50 Z100

N0070 G04 K3

N0080 M02

The processing sequence of program above is:

N0010→N0020→N0040→N0050→N0060→N0040→N0050→N0060→N0070→N0080

2.2.11 G26——Transition processing (sub-program call in the program)

Format：G26 N××××. ××××. ×××

Description: The transition processing instruction completes, the next processing section is the next one of G26 N××××. ××××. ×××, which is the difference from G25, and the rest are same as G25.

Example：

N0005 S800 M03

N0010 G26 N0050.0080.02

N0020 G4 K2

N0030 G01 X2 F20
N0040 G00 X0 Z0
N0050 G92 G90 X0 Z0
N0060 G01 Z-20 X20 F300
N0070 M00
N0080 Z-40
N0090 Z-60 X0
N0100 M02

The processing sequence of program above is:

N0005→N0010→N0050→N0060→N0070→N0080→N0050→N0060→N0070→N0080→N0020→N0030→N0040→N0050→N0060→N0070→N0080→N0090→N0100

2.2.12 G27——Endless cycle

Format：G27 N××××. ××××

Description: (1) The block between the first and second block numbers after N is the interval of endless cycle, once entering G27 status, the system will infinitely repeat carrying out the operating track which is defined by this block.

(2) In order to guaranty that the coordinate doesn't offset when every cycle starts, the block is required to be the sealed track, else the start point will shift at every start, and finally exceeds working table.

2.2.13 G28——Metric system multi-section continual thread processing command (inner-section, inter-section variable pitch)

Format：G28 Z__U__K__R__D__

Description:： Z__ length of thread
U__ taper variable
K__ thread tooth pitch
R__ pitch variable per rotation, pitch variable unit is mm/rotation
D__ thread initial angle (0~360°)

Note: (1) If multiple thread sections are programmed continually, the initial angle "D" is only valid in the first thread section.

(2) During the thread processing period, the spindle regulation switch must be kept unchanged.。

(3) The feed regulation switch is invalid.

For sample：G0 X20 Z0
G1 X19 F6000
G28 Z-20 K1
G28 Z-40 U5 K1
G28 Z-60 U5 K1
G1 X40 F6000
G0 X50 Z50

G28 typical illustrate:

Ex1 Processing of "8" oil tank

```
N0010  M03  S50
N0020  G0  X50  Z20
N0030  G0  X20.5  Z-10
N0040  G28  X19.5  Z-6  K8
N0050  G28  Z-10  K8
N0060  G28  Z-14  K8
N0070  G28  Z-10  K8
N0080  G28  X19  Z-6  K8
N0090  G28  Z-10  K8
N0100  G28  Z-14  K8
N0110  G28  Z-10  K8
N0120  G28  Z-6  K8
N0130  G28  Z-10  X20  K8
N0140  G0  X50  Z20
N0150  M02
```

Ex2 straight thread to taper thread

```
N0010  M03  S1000
N0020  G0  X50  Z20
N0030  G0  X20  Z10
N0040  G1  X19.5  F6000
N0050  G28  Z-10  K1
N0060  G28  Z-30  U5  K1
N0070  G1  X30
N0080  G0  Z10
N0090  G1  X19
N0100  G28  Z-10  K1
N0110  G28  Z-30  U5  K1
M0120  G1  X30
N0130  G0  Z10
N0140  G0  X50  Z20
N0150  M02
```

Ex2 Varying pitch thread processing in the processing

```
N0010  M03  S1000
N0020  G0  X50  Z20
N0030  G0  X20  Z10
N0040  G1  X19.5  F6000
N0050  G28  Z0  K1
N0060  G28  Z-20  K1  R0.1
N0070  G1  X25
N0080  G0  Z10
N0090  G1  X19
N0100  G28  Z-20  K1  R0.1
N0110  G1  X25
N0120  G0  Z10
N0130  G1  X18.9
N0140  G28  Z0  K1
N0150  G28  Z-20  K1  R0.1
N0160  G1  X25
N0170  G0  X50  Z20
```

2.2.14 G29——English system multi-section continual thread processing command (inner-section, inter-section variable pitch)

Format：Same as G28

Description： Thread pitch is K tooth/inch

2.2.15 G30——Magnification and minification cancel

Format：G30

Description: When the G31 magnification and minification is implemented, G30 cancels effect of G31.

2.2.16 G31——Magnification or minification

Format：G31 KX××.××

Description: (1) The scope of multiply factor is 0.001-65.5, namely K0.001-K65.5.

(2) The effect of multiply factor is to magnify or minify K times for the various segments' size of processing track evenly.

(3) The multiply factor has no effect on the cutting tool.

2.2.17 G33/G34——Metric system/English system single-tool screw thread cycle

Format: G33 Z__ K__ R__ I__ or G33 U__ Z__ K__ R__ I__

U_Z_: screw thread end point coordinate, K screw thread, R: cutting depth,

I: de-trailing length

This cycle only runs one cutting, after cycle ends, cutting tool will stop at the position after cutting feed R value in X direction, Z will return to start point of cycle, thereby realizing free cutting processing conveniently.

Example：G0 X50 Z100

```
G33 Z55 K1 R1.5 I4
G33 Z55 K1 R0.8 I4
G33 Z55 K1 R0.2 I4
……
M02
```

2.2.18 G35——Skip function

Format：G35 Z__F__

Description: After G35 command, G01 can command straight line interpolation.

(1) When this command is implemented, if the external skip signal is input, the command execution will be interrupted, and the next block will be executed.

(2) When G35Zxx occurs during the programming, the system detects the input port signal set by P69# parameter, when the signal is valid, system will set Z-coordinate for Zxx and then run the program continually; if the signal isn't detected after the programming value ends on the Z axis, system will run the program continually without processing

N0010 G0 X50 Z50

N0010 G35 Z0 F1000 (The system will detect the input port signal set by P69# parameter during this process, when the signal is valid, system will continually run the program after setting Z coordinate for 0.)

N0010 G01X52 Z-1 F150

N0010 G1Z-20

N0010 G0 X50 Z50

N0010 M02

N0010

2.2.19 G40-G42 Tool nose radius compensation

G40——Cancel tool nose radius compensation

G41——Left tool nose radius compensation

G42——Right tool nose radius compensation

Description: See description of Chapter 5 for G40-G42

2.2.20 G54——Cancel zero point bias, recover working coordinate system

Format: G54

Description: (1) After zero point is biased, G54 function will recover the programming zero of processing part to initial workpiece coordinate system when power is on.

(2) G54 function will cancel all previous coordinate bias functions.

2.2.21 G55——Absolute zero point bias

Format: G55 x__z__

Description: (1) G55 function will move the programming zero point to coordinate specified by X'O'Z' horizontally.

(2) Both X and Z coordinate can move horizontally; in addition, one coordinate can move horizontally, the zero point of un-programmed coordinate doesn't move horizontally.

(3) G55 function is independent block, and other command cant' appear in this segment.

(4) The block after G55 will be programmed based on new coordinate system established by G5 without considering influence of original coordinate system.

(5) The dynamic coordinate display in processing is still corresponding to initial coordinate system zero point.

(6) G55 isn't move command but to memorize coordinate bias, if it's necessary to move cutting tool to G54, G01 or G00X0Z0 block must be re-programmed, to move cutting tool to G54.

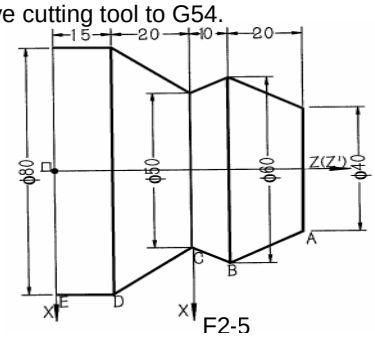
For instance: N0010 G92 G90 X40 Z65

N0020 G01 X60 Z45 F100（AB）

N0030 G55 Z35

N0040 G01 X50 Z0 （BC）

N0050 X80 Z-20 （CD）



F2-5

N0060 G54

N0070 M02

2.2.22 G56——Increment zero point bias

Format: G56 X__ Z__

Description: (1) G56 function will move the zero point of coordinate system for X'O'Z' to form new coordinate system horizontally from current position increment of cutting tool.

(2) Other cautions are same as G55.

For instance: N0010 G90 G92 X20 Z70

N0020 G01 X40 Z50 F100

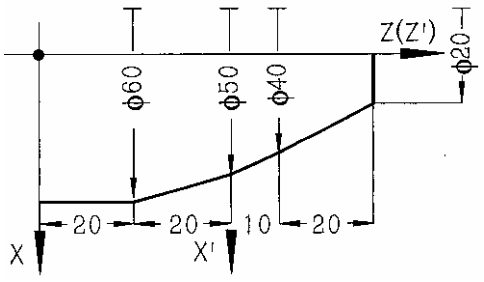
N0030 G56 Z-10

N0040 G01 X50 Z0

N0050 X60 Z-20

N0060 G54

N0070 M02



F2-6

2.2.23 G57——Bias of current point

Format: G57

Description: (1) G57 function sets the current position of cutting tool for coordinate zero point, the later programming will take this point as the coordinate zero point without considering influence of original coordinate system.

(2) The rest is same as G56.

For instance: N0010 G90 G92 X20 Z60

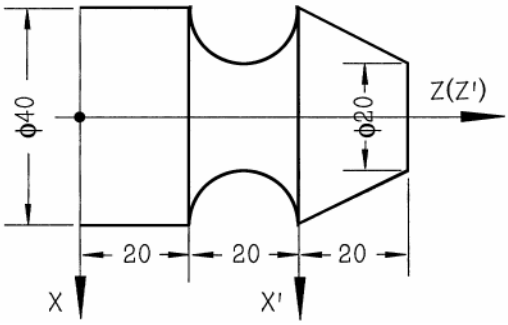
N0020 G01 X40 Z40 F100

N0030 G57

N0040 G02 Z-20 I0 K-10

N0050 G54

N0060 M02



F2-7

2.2.24 G61——Continual corner clearing of current segment and subsequent processing segment

Format: G61

Description: Both this segment and subsequent processing trace adopt sharp corner connection until they are cancelled by G64.

2.2.25 G62——Fast corner clearing command of current segment

Format: G62

Description: Sharp corner interim is between this segment of trace and the next segment (see description of 4.3.1.4)

For instance: G01 X100 Z20 F100 G62

2.2.26 G64——Cancel corner clearing interim

Format：G64

G61～G64 Description：

The cutting speed of system control between two sections of processing blocks in 4.2.1 isn't changed, the interim radian will occur between two rails, in the occasion having requirement for workpiece molding surface, these radian must be eliminated such as order shaft and various end surfaces. If G61 or G62 are adopted, the next program can be started after the cutter runs this program

completely, to ensure the shape between two mold surfaces matches the programming shape, called "Angle clear" function.

No G61 function G61 function

When the cutting is performed, because the feed speed is low, about below F300, when the cutting time constant (39＃) is smaller (<100ms), this interim radian is also small, and the influence to workpiece is also small, under the precondition of meeting processing requirement, the machining efficiency can be enhanced and the shock vibration can be reduced without adopting angle clear function, and it's favorable to enhance smoothness degree, in the occasion having higher requirement or it must be sharp corner, G61 (G62) function shall be adopted. G62 only works to current program, namely this section of program realizes angle clear, and the following program still adopts interim radian mode.G61 is valid to current section and follow-up program until G64 cancels, the interim processing is only suitable for continual G01, G02 and G03, once the follow-up program isn't be above track, system will cancel the interim function automatically.

2.2.27 G71——Inner (outer) diameter cutting compound cycle

Format: G71 I_K_N_X_Z_F

Description: The fine machining path of rough machining and fine machining showed in the instruction execution figure 2-8 is the track of A→B→C→D.

Among them: I: cutting depth (cutting amount for each time), the symbol isn't added when specified, and the direction is determined by vector AB;

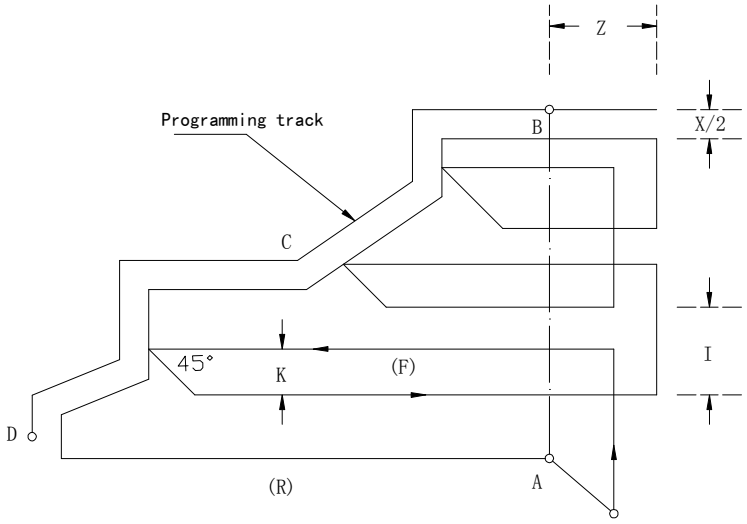
K: Retract amount for each time, the symbol isn't added when specified, and the directions of X and Z are respectively determined by X (X-direction fine machining allowance) and Z (Z direction rough machining allowance);

N: Fine machining block number;

X: X direction fine machining allowance;

Z: Z direction fine machining allowance;

F: The F in the G71 programming is valid at the time of rough machining, and the F in the fine machining block is valid at the time of fine machining.



F2-8

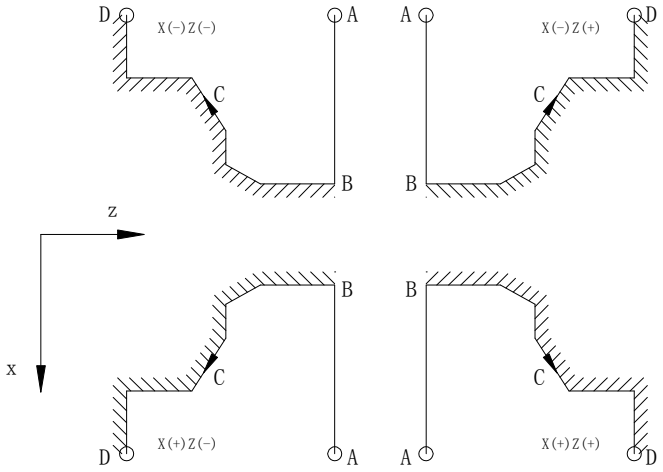
Under G71 cutting cycle, the cutting feed direction parallels to Z axis, and the symbols of X and Z are showed as Figure 2-9. (+) signifies movement along axis positive direction, and (-) signifies

movement along axis negative direction.

Note: (1) N (fine machining block number) must be larger than 1;

(2) A→B must be completed by G00 instruction, and the G00 instruction can't be included within B→C→D;

(3) The Z direction movement amount shouldn't be in the A→B block, X direction movement amount is equal to X direction movement total of B→C→D



F2-9 Symbols of X and Z under G71 compound cycle

For instance：N0010 G00 X520 Z370 F2400

N0020 G71 I14 K2 N8 X0.74 Z0.29 F2400

N0030 G00 X100 Z370

N0040 G01 X100 Z350 F240

N0050 G01 X200 Z230

N0060 G01 X200 Z170

N0070 G01 X320 Z170

N0080 G01 X320 Z100

N0090 G03 X420 Z50 I0 K-50

N0100 G02 X520 Z0 I100 K0

N0110 G00 X520 Z370

N0130 M02

%P59……Cutting internal diameter

N0010 G00 X10 Z370

N0020 G71 I7 K2 N8 X-0.37 Z0.29 F2400

N0030 G00 X240 Z370

N0040 G02 X190 Z320 I0 K-50 F240

N0050 G03 X140 Z270 I-50 K0

N0060 G01 X100 Z200

N0070 G01 X100 Z150

N0080 G01 X60 Z150

N0090 G01 X60 Z80

N0100 G01 X10 Z00

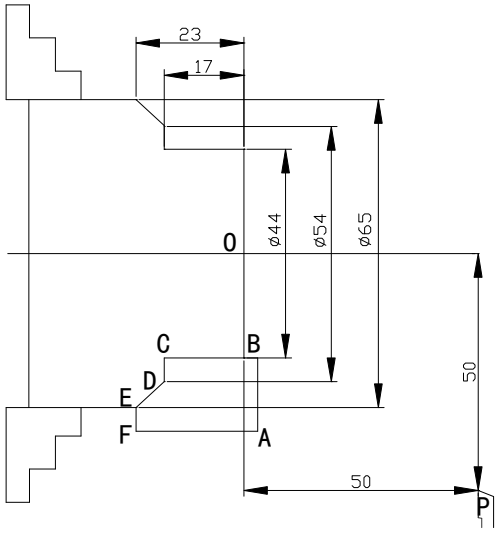
N0110 G00 X10 Z370

N0130 M02

G71 example and detailed instruction

If the 65# bar is processed to the following workpiece (O point is the coordinate zero point of

programming, P point is the cutting start point), the programming will be as follows:



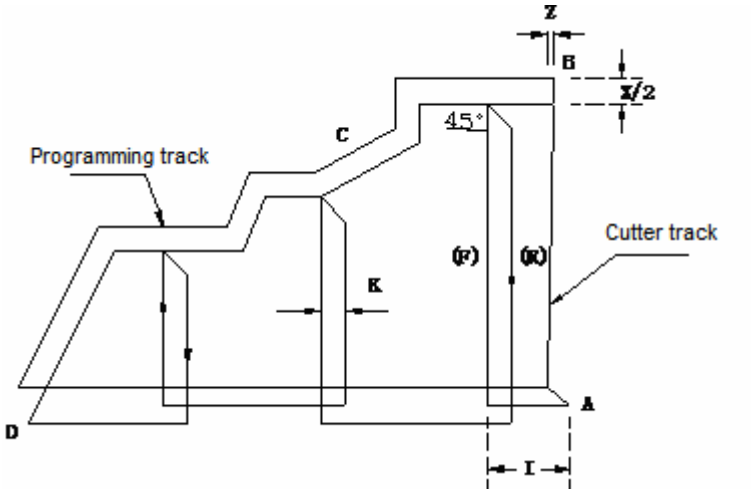
F2-10

```
N10M03S800TO1
N20G0X70Z3                ；A point, fast positioned to start point of cycle
N30G71I5K1N5X.5Z.3F800    ；N5 indicates total segments of G71 cycle (including N40, N50, N60,
                          N70 and N80), X and Z shall be programmed to value except 0
                          (remain fine turning allowance)
 N40G00X44                ；B point, must be completed by G00, Z direction can't move
N50G01Z-17F200            ；C point                --
 N60X54                   ；D point, X-direction total depth is equal to N40 segment depth (A->B)
 N70X65Z-23               ；E point, can be only interpolation commands G01, G02 and G03,
N80X70                    ；F point (fine turning end point)--M, S and T command
 N90G0X100Z50             ；Retracted to safety position (return to cutting start point P)
N100M02
```

2.2.28 G72——End surface cutting compound cycle

Format: G72 I_K_N_X_Z_F_

Description: The fine machining path of rough machining and fine machining showed in the instruction execution Figure 2-10 is the track of A→B→C→D.



F2-11 End surface cutting compound cycle G72

Among them: I: cutting depth (cutting amount for each time), the symbol isn't added when specified, and the direction is determined by vector AB;

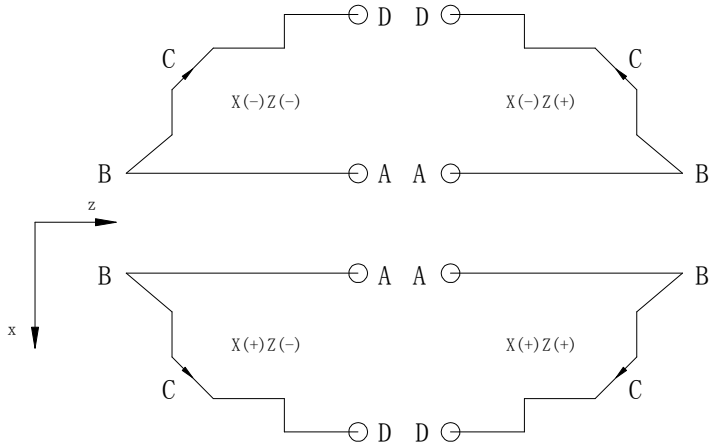
K: Retract amount for each time, the symbol isn't added when specified, and the directions of X and Z are respectively determined by X (X-direction fine machining allowance) and Z (Z direction rough machining allowance);

N: Fine machining block number;

X: X direction fine machining allowance;

Z: Z direction fine machining allowance;

F: The F in the G72 programming is valid at the time of rough machining, and the F in the fine machining block is valid at the time of fine machining



F2-12 The symbols of X and Z under G72 end surface cutting compound cycle

Note: (1) N (fine machining block number) must be larger than 1;

(2) A→B must be completed by G00 instruction, and the G00 instruction can't be included within B→C→D;

(3) The X direction movement amount shouldn't be in the A→B block, Z direction movement amount is equal to Z direction movement total of B→C→D

Example：N0010 G00 X520 Z370

N0020 G72 I7 K2 N8 X0.37 Z0.29 F2400

N0030 G00 X520 Z40

N0035 G01 X500 Z90 F240

N0040 G01 X320 Z140

N0060 G01 X320 Z210

N0070 G01 X200 Z210

N0080 G01 X200 Z270

N0090 G03 X100 Z320 I0 K50

N0095 G02 X0 Z370 I-100 K0

N0100 G00 X520 Z370

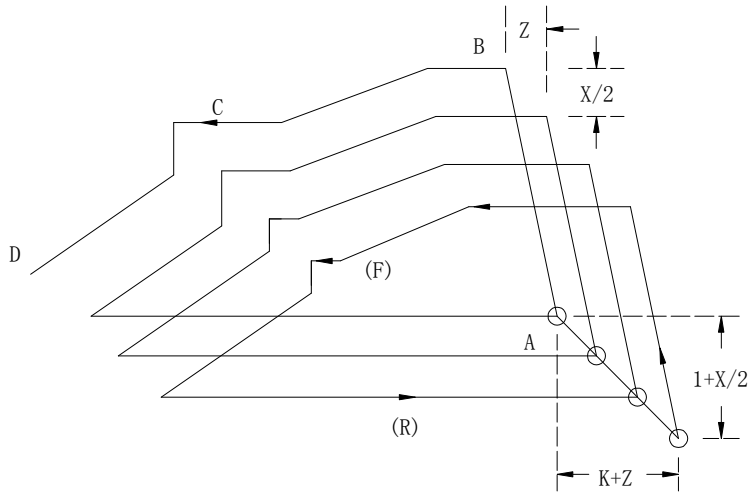
N0120 M02

2.2.29 G73——Sealed contour compound cycle

Format：G73 I_K_N_L_X_Z_F_

Description: The cutting tool track showed in Figure 2-13 is the closed loop when this function is of cutting work, and the cutting tool feeds gradually, which makes the closed cutting loop gradually approach to the element final shape and finally cut into the shape of work piece, and its fine machining path is the track of A→B→C→D.

This instruction can implement highly-efficient cutting to the preliminary-finished work piece in the rough machining such as casting, forging and so on.



F2-13 Sealed contour compound cycle G73

Among them:

I: The rough machining total allowance in the X direction;

K: The rough machining total allowance in the Z direction;

N: Fine machining block number;

L: Rough cutting number;

X: X direction fine machining allowance;

Z: Z direction fine machining allowance;

F: The F in the G73 programming is valid at the time of rough machining, and the F in the fine machining block is valid at the time of fine machining

Note: I and K signify the total cutting amount of rough machining, and the rough cutting number is L, then the cutting amounts in the X and Z directions for each time are I / L and K/L;

Example：N0010 G00 X540 Z390

N0020 G73 I440 K60 N6 L20 X0.6 Z0.3 F2400

N0030 G00 X80 Z370

N0040 G01 X80 Z270 F240

N0050 G01 X150 Z140

N0060 G02 X350 Z40 I200 K0

N0070 G01 X400 Z20

N0080 G01 X520 Z00

N0090 G00 X540 Z390

N0110 M02

Through utilizing this group of compound cycle command, it's only necessary to appoint fine machining route and engagement of rough machining, and system will automatically calculate rough machining route and cycle number.

2.2.30 G74——Returning to reference point (mechanical origin)

Format：G74 X_Z_

Description: (1) Other contents in this segment are prohibited.

(2) The coordinates following G74 will return to zero in turn with X, Z sequence.

(3) It is must be confirmed that the reference point switch to be installed on the machine tool before using G74

2.2.31 G75——Returning to presetting cutter point from reference point

Format：G75 X_Z_

Description: (1) Other contents in this segment are prohibited.

(2) After G75 instruction execute，X axis and Z axis move to the machine tool coordinate XP, and ZP is the coordinate position set by P8# and P9# parameter.

(3) After G75 instruction completes, Recover workpiece coordinate (global coordinate) of X axis and Z axis to the value set by P18# and P19#.

(4) B033=1 (zero return mode isn't required in power-on), G75 will be invalid.

(5) G75Y will move to machine tool coordinate in Y axis, YP is the coordinate value set by P24#, and recover the PB4# set value to the workpiece coordinate of Y axis.

(6) G75 function must ensure that the machine tool coordinate and workpiece coordinate at start position is the coordinate of actual cutting tool at this position.

2.2.32 G76——Returning to processing start point from current position (feed point)

Format：G76 X_Z_

Description: (1) Other contents in this segment are prohibited

(2)The coordinate relative to origin on the machine tool is displayed with big coordinate, the cutting tool position coordinate of processing start can be memorized in P18# and P19# parameter.，This function can return to this position from machine tool optional position, and the speed is same as G00.

(3) (3) The processing start point (P18# and P19#) is set referring to processing origin (like chuck center), and the result of G76 execution is to move tool nose to the same coordinate position with parameters P18#and P19#.

(4) The workpiece coordinate of cutting feed point corresponding to Y axis is PB4# (P114#).

2.2.33 G79——-Metric system end surface thread cycle.

Format: G79X(U)__ K__ I__ R__ L__

Cycle number

Depth of thread

De-trailing length in Z direction after the. thread is cut

Thread pitch (Metric Kmm, English system K tooth/inch)

Thread end X-axis coordinate. Absolute and relative coordinate programming can be adopted

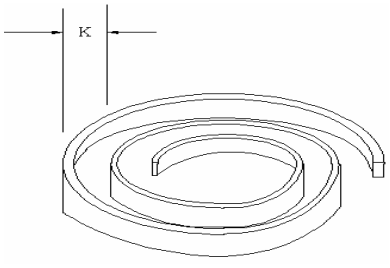
Note: (1) The parameters set in the end surface thread processing is same as G86 straight thread, see 2.2.38 Paragraph for detailed description.

(2) End surface thread non-single tool cutting.

(3) End surface thread no screw-in cutting function.

Example

```
N0010   G0 X 100   Z 100
N0020   G0 X 50   Z 0
N0030   G79 X 0 K2 R1 I4 L6
N0040   G0 X 100 Z 100
N0050   M02
```



F2-14

2.2.34 G80-English system end surface thread cycle.

Format: Same as G79.

Note: Thread pitch is K tooth/inch

2.2.35 G81——Excircle (inner circle) fixed cycle.

Format：G81 X_Z_R_I_K_F_

Description:

(1) Under absolute coordinates mode, X and Z are absolute coordinates of another end surface (end point), and under relative value programming mode, X and Z are increment values of end point which is relative to current position.

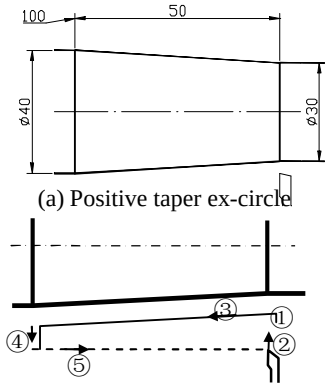
(2) R is the processing diameter of start point section.

(3) I is the rough turning feed amount and K is the fine turning feed amount. I and K are symbolic numbers, and their symbols should be the same. The symbols are specified as follows: Cutting from outward central axis (turning ex-circle) is "-", on the contrary it's "+".

(4) Various X, Z and R values determine different shapes of ex-circle such as: with taper or without taper, positive taper and negative taper, left cutting or right cutting and so on.

(5) F is the feed speed (mm/min) of cutting processing.

(6) After processing completes, the cutting tool stops at the end point.



(a) Positive taper ex-circle

(b) Processing flow chart

F2-14

Example 1: positive taper ex-circle, the left cutting is implemented

G90 G81 X40 Z100 R30 I-1 K-0.2 F200 (Absolute value programming)

G91 G81 X0 Z-50 R30 I-1 K-0.2 F200 (Relative value programming)

Processing course:

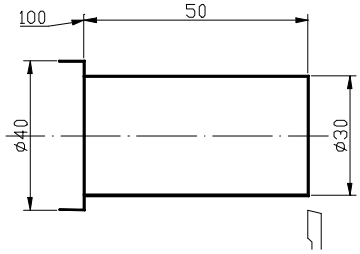
G01 feeds two times of I (the first cutting is I, and the final cutting is I+K fine turning), implementing depth cutting;

G01 two axes interpolation, cutting to the end point section, and it will stop if processing completes;

G01 retracts I to safety position, and the auxiliary section smoothness processing is implemented simultaneously.

G00 retracts △Z to start point section;

G00 fast feeds to I position off from work piece surface, and I is preserved to implement the next step cutting processing and repeated to ①.



F 2-15 Ex-circle without taper

Example 2: Non-taper ex-circle, the left cutting is implemented

G90 G81 X30 Z100 R30 I-1 K-0.2 F200 (Absolute value programming)

G91 G81 X-10 Z-50 R30 I-1 K-0.2 F200 (Relative value programming)

Example 1 of processing course

Example 3: Negative taper ex-circle, the left cutting is implemented

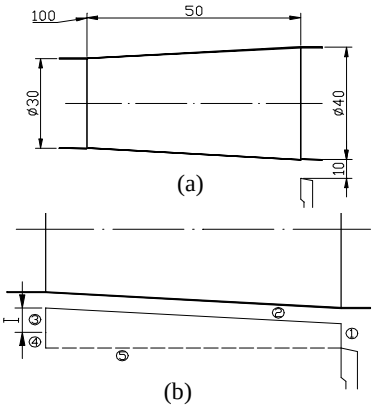
Taking into consideration of cutting amount of end point, the cutting tool should maintain proper distance (≥△Φ) from work piece at the start point.

G90 G81 X30 Z100 R40 I-1 K-0.2 F200 (Absolute value programming)

G91 G81 X-30 Z-50 R40 I-1 K-0.2 F200 (Relative value programming)

Processing course:

G01 feeds two times of I (the first cutting is I, and the final cutting is I+K fine turning), implementing depth cutting;

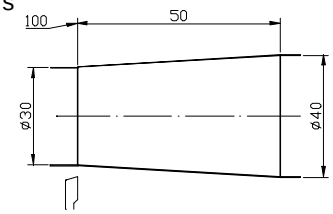


(a)

(b)

F2-16

G01 two axes interpolation, cutting to the end point section, and it will stop if processing completes;
G01 retracts I, and the auxiliary section smoothness process is implemented s
G00 fast retracts △Φ to safety position;
G00 fast retracts to start point section and is repeated to ①.



Example 4: Negative taper ex-circle, the right cutting is implemented

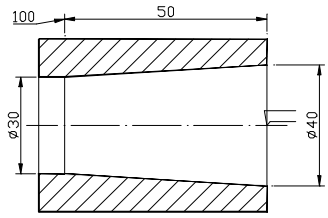
Absolute value programming:

G90 G81 X40 Z150 R30 I-1 K-0.2 F200

Relative value programming:

G91 G81 X0 Z50 R30 I-1 K-0.2 F200

The processing course is same as example 1



F2-18

Example 5: Negative taper inner circle, the left cutting is implemented

G90 G81 X30 Z100 R40 I1 K0.2 F200 (Absolute value programming)

G91 G81 X30 Z-50 R40 I1 K0.2 F200 (Relative value programming)

Processing course:

It's similar to example 1, and the difference is that the cutting tool retracts to the central axis direction when retracting

2.2.36 G82——End surface fixed cycle.

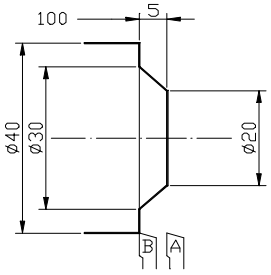
Format: G82 X_Z_R_I_K_F_

Description: (1) Under absolute coordinates mode, X and Z are absolute coordinates of another end surface (end point), and under relative value programming mode, X and Z are increment values of end point which is relative to current position.

(2) R=(end point diameter—start point diameter),the end point (start point) diameter is the final cone diameter on the end point (start point) section. When the flat end surface cutting short work piece processes, the end-point diameter and start point diameter are both zero. R is the symbolic number, "+" signifies that end-point diameter is greater than start point diameter, and "-" is the opposite.

(3) I is the rough turning feed and K is the fine turning feed. I and K are symbolic numbers, and their symbols should be the same. The symbols are specified as follows: leftwards cutting processing is "+" (omission is allowable), and rightwards cutting processing is "-".

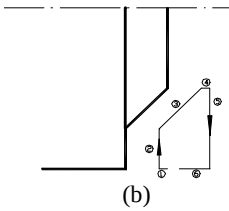
(4) Various X, Z, R and I values determine different shapes of end surface, among them R value determines whether there is taper, and end surface has no taper when R is equal to 0; If X=0 and R=0 at the same time under absolute mode, then the work piece will be cut short and end surface is turned flat; the symbol of R determines taper direction with taper end surface; all symbols of Z, R and I determine the cutting mode of taper end surface, which is divided into internal cutting and external cutting. Towards to programming of various conditions, they will be elaborated on with examples.



(a)

(5) F is the feed speed (mm/min) of cutting processing.

(6) Towards to the end surface without taper, the processing length has no limit; but towards the end surface with taper, the processing length is limited within the one between two end surfaces, and cutting tool requires stopping on one of two end surfaces before processing starts. After processing completes, the cutting tool stops at the programming end point.

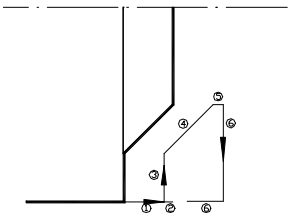


(b)

Example 1: The end surface cycle which taper has steps and the cutting tool can stop at A or B position.

Start from A position.

G90 G82 X30 Z100 R10 I-1 K-0.2 F200 (Absolute value mode)



(c)

F2-19

G91 G82 X0 Z-5 R10 I-1 K-0.2 F200 (Relative value mode)

Processing course:

G01 feeds two times of I in Z direction (the first cutting is I, and the final cutting is I+K fine turning), G01 feeds to the end point in X direction to implement depth cutting;

G01 implements two-axis interpolation, cutting to the other end surface; G01 retracts I to safety position in Z direction, and the auxiliary section smoothness process is implemented simultaneously;

G00 retracts to start point in X direction;

G00 fast feeds to I position off from work piece surface in Z direction, and I is preserved to implement the next step cutting processing.

If the processing completes, G01 will feed to end point and stop, else it'll repeated to ①.

Start from B position

G90 G82 X20 Z105 R-10 I-1 K-0.2 F200 (Absolute value mode)

G91 G82 X-20 Z5 R-10 I-1 K-0.2 F200 (Relative value mode)

Processing course:

G00 fast feeds to A position in Z direction ;

G01 feeds two times of I in Z direction (the first cutting is I, and the final cutting is I+K fine turning), implementing length cutting;

G01 feeds to the end point in X direction to implement depth cutting;

G01 implements two-axis interpolation, cutting to the other end surface;

G01 retracts I to safety position in Z direction, and the auxiliary section smoothness processing is implemented simultaneously.

If the processing completes, G01 will feed to end point and stop, else G00 will retract to start point in X direction;

G00 fast feeds to I position off from work piece surface in Z direction, and I is preserved to implement the next step cutting processing.

Repeated to ②.

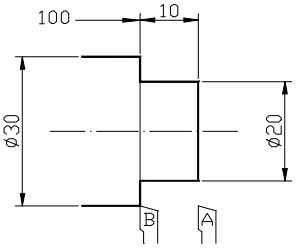
Example2：The end surface cycle which taper has no steps and the cutting tool can stop at A or B position.

Start from A position.

G90 G82 X20 Z100 R0 I-1 K-0.2 F200 (Absolute value mode)

G91 G82 X-10 Z-10 R0 I-1 K-0.2 F200 (Relative value mode)

Processing course: It's similar to example 1.A, and the difference is without the ③ step.



F2-20

Start from B position

G90 G82 X20 Z110 R0 I-1 K-0.2 F200 (Absolute value mode)

G91 G82 X-10 Z10 R0 I-1 K-0.2 F200 (Relative value mode)

Processing course: It's similar to example 1.B, and the difference is without the④ step.

Example 3: The end surface cycle which taper has no steps, the cutting tool can stop at A or B position.

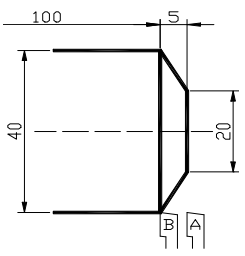
Start from A position.

G90 G82 X40 Z100 R20 I-1 K-0.2 F200 (Absolute value mode)

G91 G82 X0 Z-5 R20 I-1 K-0.2 F200 (Relative value mode)

Processing course: It's similar to example 1.A, and the difference is without the ② step.

Start from B position



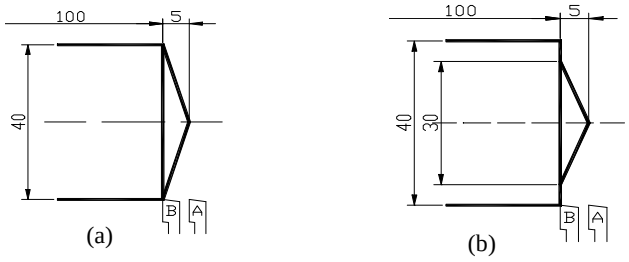
F2-21

G90 G82 X20 Z105 R-20 I-1 K-0.2 F200 (Absolute value mode)

G91 G82 X-20 Z5 R-20 I-1 K-0.2 F200 (Relative value mode)

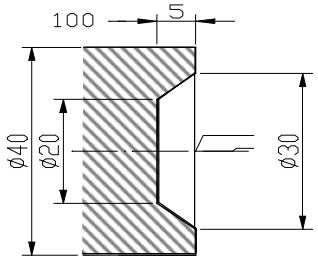
Processing course: It's similar to example 1.B, and the difference is without the③ step.

According to various X and R values, the following figure can also be programmed:



F2-22

Example 4: The concave end surface cycle with steps. The cutting tool should stop at the central position when it's processed from center to outside, which guaranties the cutting tool not to be bumped at the time of retracting.



F2-23

G90 G82 X20 Z100 R-10 I-1 K-0.2 F200 (Absolute value mode)

G91 G82 X20 Z-5 R-10 I-1 K-0.2 F200 (Relative value mode)

In the examples above, if I and K are changed into "+" simultaneously, then the figure will regard X-axis as mirror image and right processing will be implemented.

2.2.37 G83-Deep hole processing cycle

It's used for interval feed of Z axis. Rapidly retract to the top position of hole after each feed cutting and pause the second set by PAO# parameter (favorable for chip removal), later move to the position to the previous hole bottom with K distance. Perform second cutting feed cycle until cutting to the hole bottom represented by I. Pause the second specified by R, finally, rapidly retract to the top position of hole, and G83 instruction segment is completed.

Format1：

G83 X__ Y__ Z__ I__ J__ K__ R__ F

Format2：

G83 X__ Y__ Z__ D__ I__ J__ K__ R__ F

The difference from format 1 is that the D parameter is used to appoint the cut depth of the first cutter (greater than J)

Z: Coordinate of hole top

I: Coordinate of hole bottom

J: Feed depth every time (unsigned number)

K: The distance from previous hole bottom (unsigned number) from fast feed to work feed when performing second feeding after retracting every time

R: Hole bottom delay time

D: Cut depth of first cutting (unsigned number)

F: Feed speed

Example 1：G92 X60 Z130

M03 S500

G90 G83 X100 Z90 I30 J20 K10 R1 F600

M02

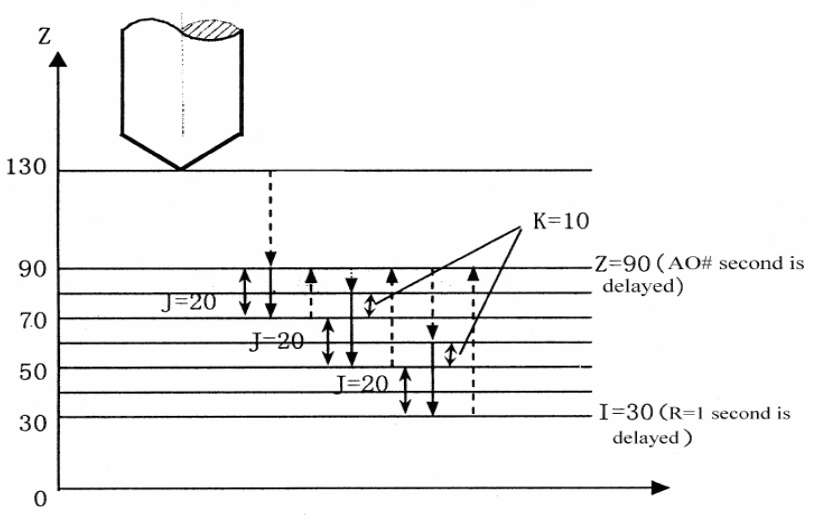
Example 2：G92 X60 Z130

M03 S500

G91 G83 X40 Z-40 I-60 J20 K10 R1 D30 F600

M02

G83 command operation cycle is as Figure 2-23:



F2-24 G83 command operation diagram

2.2.38 G84-Metric rigid threading cycle

Format: G84 Z__ K__ L__ N

Description: (1) G84 (G85) can be only used in case of installing the spindle encoder.

(2) Z is the threading terminal coordinate, K is the pitch.

L: Material compensation amount, scope is from 0 to 15; 0 is for general material (L value isn't programmed), L can be increased for fragile material to enhance the threading rotation speed.5 to 10 can be properly increased for sticky material to reduce possibility of broken threading. If L isn't programmed, it'll be determined by P87#.

N: After threading feed reaches Z value, the system will give spindle stop signal; when the spindle reduces to the rotation speed set by N, the system will give reverse signal, thus reducing the reversing time; when N isn't programmed, the system will give spindle reverse signal after checking that the rotation speed of spindle is reduced to 0.

When the spindle is under frequency control, it'll be useless to program N owing to characteristic of frequency converter.

(3) Select rotation speed of spindle in rigid threading.

When rigid threading is performed, one thread pitch will be fed axially along the spindle every time the spindle rotates for one circle, and this relationship should be strictly maintained when the rotation speed of spindle is increased and decreased. The spindle multiplying factor and feed multiplying factor are prohibited in threading.

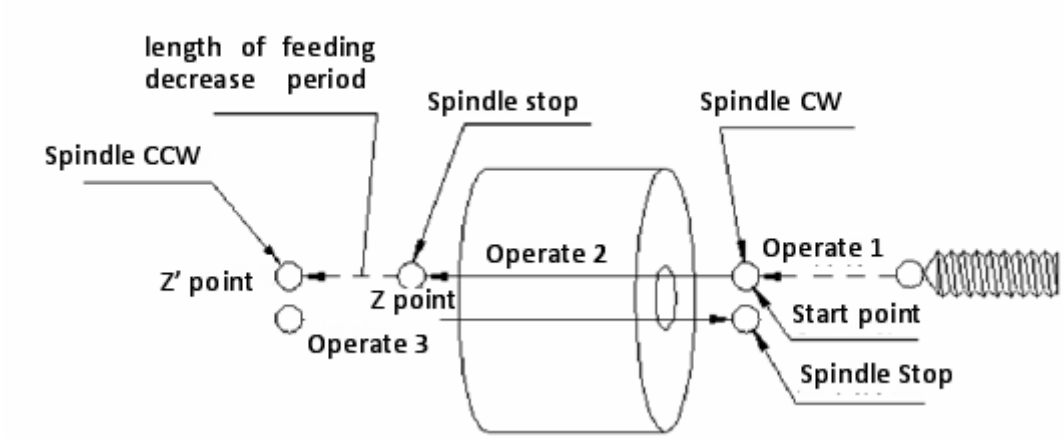
When rigid threading is performed, the Z-direction feeding is synchronized with the spindle; when threading feed reaches Z value, the system will give spindle stop signal; within the deceleration time for spindle from the set rotation speed to zero speed, Z direction is still followed up (the longer the time of spindle deceleration time is, the bigger the follow-up length is); in order to reduce the follow-up length within the deceleration time, the rotation speed increase and decrease time should be reduced as possible.

The proportional relation between the threading feed speed and the spindle speed is as follows:

F=S×K (Formula 2-1)

In the formula:

F-Threading feed speed; S-Spindle speed; K-Tap thread pitch;



F2-25

Description:

Operation 1: Rapidly position to the start point of threading and the spindle rotates positively.Operation 2: Threading feeds to Z point and spindle stops.Operation 3: Spindle rotates negatively, tap returns to the start point and spindle stops

2.2.39 G85-English system rigid threading cycle

Format: Same as G84.

Description: Thread pitch is K tooth/inch.

2.2.40 G86——Metric thread cycle

格式： G86 X Z K I/D R N L J

- J: Z direction de-trailing length (Plus positive value)
- L: Cycle number
- N: Thread head number, which is used for multiple thread (N≤150)
- R: The diameter difference between thread outside diameter and root diameter, which is positive value
- D: The de-trailing length after thread is completed in X direction/screw in distance value
- I: Thread pitch (metric system Kmm, English system K teeth/Inch)
- K: Length of thread, it can be absolute or relative programming
- X: X is the diameter change in x direction, and it's the straight thread when x is equal to 0

Description:

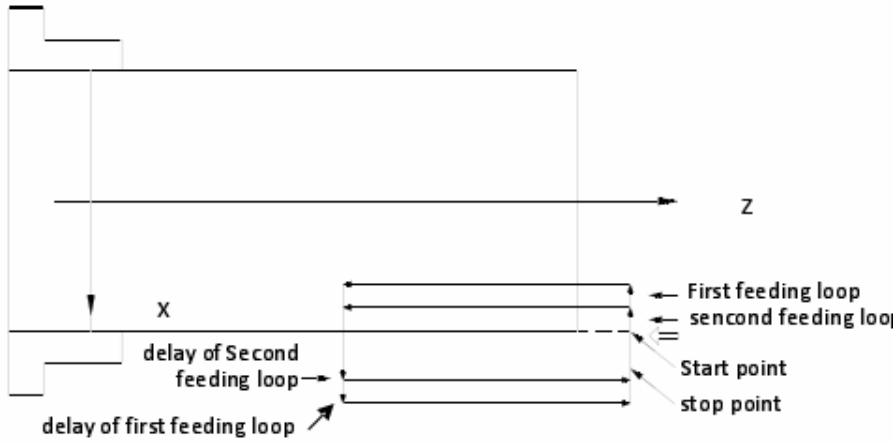
The macro variables P10, P11 and P12 are especially for setting other parameters of thread processing.

(1) The feed depth of each time is determined by P10 and P11assignment statements before program, and the X direction single side feed smooth thread flat of final cutting.（smoothing tool amount is determined by P25# parameter)

（2）The de-trailing direction of thread in X direction is determined by I value, "+" is the external thread and "-" is the internal thread.

（3）The initial position of thread processing cycle is where the tool nose is aligned with thread top diameter.

(4) J value indicates amendment of Z direction de-trailing position (see description later).



F2-26

(5)When it's necessary to wait for thread pitch ending, the screw out function can be used, and it's format is to add J value into the general G86 function. When J value isn't compiled, the de-trailing will be implemented after Z direction moves to approach end-point.

(6) Normally the de-trailing length is signified with value after I in X direction of G86, when D is compiled, it signifies that the thread feed is screw in. Please note when using screw in function: When feed is started, the tool nose must be in the distance greater than or equal to D value outside of work piece surface, else the cutting tool will be bumped; the screw in distance is equal to de-trailing one (X direction).I value isn't programmed and used to process the fixed point retracting of small hole thread, and the X retracting direction is determined by the sign of R

(7) According to I, J and D programming The general combination is as follows：

a）G86 Z-100 R2 K3 L10 I5 common thread processing

b）G86 Z-100 R2 K3 L10 I5 J6 Retract by 6mm ahead of Z-direction normal de-trailing position

c）G86 Z-100 R2 K3 L10 D5 screw in cut-in but without equal thread pitch de-trailing

d）G86 Z-100 R2 K3 L10 D5 J6 Retract by 6mm ahead of Z-direction normal de-trailing position of screw in and screw out

e）G86 Z-100 R-2 K3 L10 External thread fixed point retracting cycle cutting

f）G86 Z-100 R2 K3 L10 Internal thread fixed point retracting cycle cutting

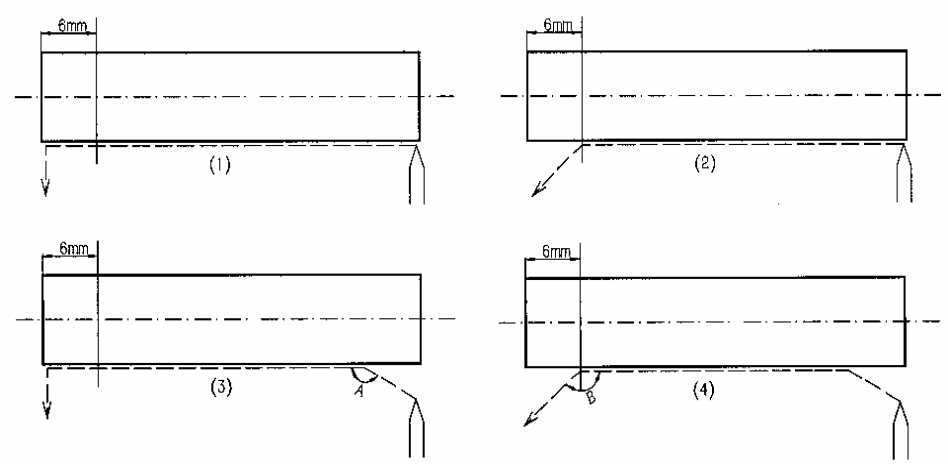
g）G33 Z-100 R-2 K3 Single-tool external thread cutting

h）G33 Z-100 R2 K3 Single-tool internal thread cutting

he user can set various programming parameters flexibly according to specific circumstances.

The screw-in cut-in angle and cut-in position can be provided with different angles according to the thread processing speed (spindle speedXK thread pitch), X-direction screw-in and screw-out time constant and speed (P16#, P49#, P59#), Z-direction time constant and speed in thread processing (P40#, P45#).

(8) The 16# parameter signifies the screw in and screw out speed in X direction, and the general programming value is from 200 mm／ min to 6000 mm /min. However when the set value is less than 100 mm/min, it'll be automatically set to 2500 mm/min when system processes.



F2-27

(9) According to the cut depth of each cut in thread cutting，it can be optionally set in the program, adopting the assignment statement P10=0, 1 and 2. A. When P10=0, it's the equidistance feed at the time of thread cutting, namely the feed amount of each time is R／L, and when the 25# parameter is not equal to 0, finally increasing one cut of smoothing tool(calculate the general cutting output R)

B. P10=1, perform equal cutting output feeding. For 60 degree screwer, the metal cutting output of each feeding can be guaranteed basically same. The cutting output of each cut is ΔRn=（$\sqrt{n}$ -$\sqrt{n-1}$）×R/$\sqrt{L}$,Rn: Feed amount of the n time. N: Feed of the n time. N≤L. L: Cycle number, R: Total cutting depth. C. When P10=2, it's the equal cutting amount feed, and the first cut when P10=1 is divided into two cuts for cutting. If the cutting amount of first cut is too big, P10 can be set as 2, and system will divide the first cut into two for cutting, to prevent damaging tool nose such as when R=1.0, L=5:

P10	The first cut	The second cut	The third cut	The fourth cut	The fifth cut	The sixth cut	The seventh cut
0	0.2	0.2	0.2	0.2	0.19	0.01..	/
1	0.45	0.19	0.14	0.12	0.09	0.01	/
2	0.23	0.22	0.19	0.14	0.12	0.09	0.01

（10）Towards to the thread cutting mode, it can be optionally set in the program, adopting the assignment statement P11=0, 1, 2 and 3

P11=0: Blade cuts in along the screw thread tooth form midline

P11=1: Blade cuts in along the left of screw thread tooth form

P11=2: Blade cuts in along the right of screw thread tooth form

P11=3: Blade cuts in along the left and right of screw thread tooth form in turn

Example：N0010 P10=2 P11=0

N0020 G00 X100 Z100

N0030 G00 X50 Z1

N0040 G86 Z50 K1 R1 I6 L5

N0050 G00 X100 Z100

N0060 M02

If the feed method can't meet the requirements, the G33 single-tool screw thread cycle customized cut depth can be adopted.

(11) There is the course of speed increase and decrease when the thread processing is started and completed, and the thread is inaccurate within this period, accordingly these two regions must be avoided at the time of actual processing. The P40# and P45# parameters define the adjustable speed increase and decrease length of Z direction acceleration when the thread is processed.

The speed of stepping motor/servomotor shouldn't exceed certain value when the thread is processed, such as 4m/min, and this speed is related with machine tool size and power of motor.

(13) CNC measures the spindle rotating speed before thread is cut, and determines the optimal course of speed increase and decrease of stepping motor, as well as judges whether spindle rotating speed is stable, after the zero signal of encoder appears, the processing will be started, and this course requires 50—100 millisecond. If the spindle rotating speed is not stable, CNC will start processing after the spindle rotating speed is stable. If the steady speed isn't measured, the thread processing won't be implemented in general. 23# parameter signifies percentage of spindle speed fluctuation, which 5-15 is used in normal, and the spindle rotating speed fluctuation ratio should be less than or equal to 23# parameter at actual processing.

(14) The 25# parameter sets the smoothing tool amount of the final cut of thread, and if thread processing cycle requires no smoothing tool finally, the 25# parameter should be equal to 0.

Note: At screw in (D value is valid), only straight thread can be processed. J must be positive value

(15) Fixed point retract

When the small hole internal thread is processed, I value can't be defined owing to retracting space, and X direction can only retract to the processing start position.The retract direction in X direction is determined by the sign of R.

Format:

G86 Z-30 K1 R1.5 L3 （Internal thread）

G86 Z-30 K1 R-1.5 L3 （External thread）

(16) De-trailing ahead of time and de-trailing with time delay (J value)

J value isn't programmed, de-trailing is normal in X direction, and the de-trailing position occurs when the speed reduction in Z direction starts.

When J is greater than 0, the X-direction de-trailing position is Jmm ahead of normal de-trailing position.

Description: When J value isn't programmed, X-direction de-trailing occurs when Z coordinate is 50.If the programming value of J is 3, X-direction de-trailing occurs when Z coordinate is 53.

When J is less than 0, J is a percentage, the position of starting de-trailing in X direction is back of the Z-direction normal position, and the back length is equal to J percent of Z-direction de-trailing length.

J=0, it's equal to normal de-trailing position.

J=-100, it's equal to de-trail when the screw is processed to the whole length (root of screw).

0≤J≤100, X-direction de-trailing point occurs between the root of screw and Z-direction normal de-trailing position.

(17) Single-tool screw thread

When the thread feeding uses equal distance one or equal cutting output one but the requirements can't be met, the single-tool thread cycle G33/G34 can be adopted to manually set cut depth every time (G33/G34 are respectively metric/English system)

Please see the program below (M45X1 external thread)

```
G0 X50 Z100                ；To the end surface of thread
G33 Z55 K1 R-6.5           ；Thread cuts in 1.5mm
G33 Z55 K1 R-7.3           ；Thread cuts in 0.8mm
G33 Z55 K1 R-8.5           ；Thread cuts in 1.2mm
G33 Z55 K1 R-8.7           ；Thread cuts in 0.2mm smoothing tool
……
G0 X70
M02
```

Note: a) When single-tool thread cycle is used, the change of spindle speed between two segments of G33 shouldn't be greater than P23#.

B) After the Z-direction acceleration (P40#, P45#) is modified, new workpiece must be used for new processing.

(18) Improve thread quality:

A) Flexible treatment:

The system sets P80#KP, P81#KI, P82#KD parameters which can improve the follow-up performance of thread processing and operating quality of X and Z direction motor.

Only KP (P80#) is adopted for the system control mode of pulse output, and KI and KD should be set for zero.

B091=1, KP is valid; KP≤100, the smaller the value is, the better the stability of thread processing is, but the micro pitch error is increased a little (no pitch accumulated error); when the pitch isn't changed, the spindle speed can be enhanced to enhance the smoothness degree of spindle; KP shouldn't be too small, KP is equal to 75 to 100 in general; when KP is equal to 100 and the processing requirement can be met, it's suggested not to use KP, namely B091 is set for 0.

b) Fast de-trailing: Through adjusting the spindle speed, X and Z direction acceleration and J value, the thread end shape with various quality can be implemented.

Increase P45# and decrease P40# to enhance the Z-direction acceleration, and the speed increase and decrease length of thread pitch is reduced and the effective thread length is increased.

Increase P16#, P49# and decrease P59# to delay X-direction de-trailing time, which is favorable to improve the tooth depth uniformity of the thread end.

The quality of thread de-trailing can be improved through changing J value and sign: when J is greater than 0 and X-direction acceleration is decreased, the ending that several circles are gradually fewer; when J is less than 0 and X-direction acceleration is increased, the short ending slot can be formed by properly reducing Z-direction acceleration.

(19) List of parameters related to thread processing;

a) P59#: Upper limit of thread X-direction screw in/screw out speed, acceleration is calculated combining P49#, and the operation of screw out is same as de-trailing.

b) P20#: Encoder line number

This parameter must be matched with each phase and each rotation of the spindle encoder installed on the machine tool, else the processing pitch will be incorrect and the processing quality will be influenced. The pulse of system to encoder is 4 multiple-frequency, and 1200 line encoder system will check 4800 lines.

c) P25#: Smoothing tool amount of final cut of thread

The thread X-direction feeding is completed for several times (determined by L), when P25#≠0, each cut feed in X-direction is decreased by P25# from R value, then cutting will be performed for L times, finally perform smoothing tool processing once and the whole cycle number is L+1

d) Upper limit of Z-axis time constant and speed when P40# and P45# thread are processed

Calculate Z-direction acceleration combining other value but not to confirm Z-direction processing speed according to this.

e) Upper limit of X-direction screw in and screw out time constant and speed when P49# and P59# thread are processed

When the thread without in-and-out cutter slot is processed, determine X cut-in and de-trailing acceleration

f) P88#: Actual speed of thread in X-direction feeding

Actual speed based on G01 moving cut-in thread when X-axis cuts in thread.

g) P84#: 1# thread tooth sharp corner

P85#：2# thread tooth sharp corner

The normal thread cutter is 60° sharp corner, when the processing thread adopts single side cutting (P=1, 2, 3, and the tooth sharp corner ≠60°).The tooth sharp corner of thread must be defined by P84# and P85#.Meaning of P84# and P85#: P11≠0 (cut along the thread single side)

P12=0：60° tooth sharp corner

P12=1：55° tooth sharp corner

P12=2：65° tooth sharp corner

P12=3： tooth sharp corner defined by user and determined by 84# parameterP12=4 ： tooth sharp corner defined by user and determined by 85# parameter

P84# and P85# are the one that the tangent value of tooth sharp corner half angle is multiplied by 10000. For instance: 62° tooth sharp corner

$$84\#/85\#=10000\times tg\frac{62°}{2}=6009 \qquad \textbf{(Formula 2-2)}$$

For instance: Process one tooth sharp corner which is 62°, and the cutter edge cuts in along the left of tooth form (set P85# for 6009)

N0100 P11=1 P12=4

N0110 G86 Z50 K2 I4 R1 L3

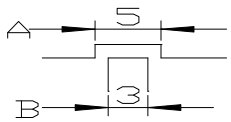
If the standard 60° thread cutter is adopted, or the single side cutting mode isn't adopted, P12 macro variable won't be required.

Programming description of increasing T-type thread

a) Before the thread program is programmed, the tooth form of T-type thread must be assigned a value with assignment statement

b) P12=5 (T-type thread processing)

c) P0=A-B (thread bottom width－tooth sharp width) ， P0≦2 times of cutter width



4. When the system parameter P25=0, it's suggested that the thread cycle number L=odd number such as L7 and L9 and so on.

5. When the system parameter P25≠0 (smoothing tool amount≠0), it's suggested that the thread cycle number L=even number such as L8 and L10 and so on.

6. Programming example:

a: thread bottom width A=5,

b: tooth sharp width B=3,

c: system parameter P25=0.1（smoothing tool amount）

```
N0010  P12=5 P0=2
N0010  M03 S1000 T1
N0010  G0 X100 Z50
N0010  G0 X30 Z10
N0010  G86 Z-50 K6 R3 I8 L10
N0010  G0 X100 Z50
N0010  M02
N0010
```

7. T-type thread non-equal cutting output processing

The first cut of thread is processed along the middle of slot width, from the second cut, the right and left

slot edge are processed in turn; when P25=0, the thread cycle number is L time, P25≠0, the thread cycle number is L+1 time. There is no feed in X direction for the final cut, to ensure that the tooth bottom has the same depth.

Cutting output at both sides of T-type slot:

1、 P25=0, the cutting output of T-type slot of every side each time is P0/L-1

2、 P25≠0, the cutting output every time is P0-P25/L-2

The T slot cutting output of the final two cut is P25, and the X-direction feeding is that the last but one cut X-direction feeding is P25, and the final one cut X-direction feeding is 0.

2.2.41 G87—English system thread cycle

Format: Same as G86. Note: The thread pitch is K teeth/Inch

2.2.42 G90—Programming with absolute value mode

Format: G90

Description:（1）When G90 is compiled into program, the programming zero will be regarded as benchmark for all the compiled coordinate values afterwards.

（2） After the system is powered on, the machine tool is in the G90 status

Example: N0010 G90 G92 X20 Z90

N0020 G01 X40 Z80 F100

N0030 G01 Z60 F50

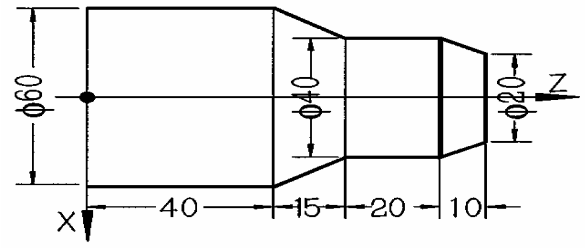
N0040 G03 X60 Z50 I0 K-10

N0050 M02

2.2.43 G91—Programming with incremental mode

Format：G91

Description: When G91 is compiled, all the coordinate values afterwards will be based on the previous coordinate position as the start point to calculate movement programming value. In the following coordinate system, the previous point will always be regarded as the start point for programming.



F2-28

Example: N0010 G91 G92 X20 Z85

N0020 G01 X20 Z-10 F100

N0030 Z-20

N0040 X20 Z-15

N0050 M02

2.2.44 G92—Setting work piece coordinate system

Format: G92 X_Z_

Description: (1)G92 only changes the current displayed coordinate value of system, and the coordinate axes require no shift to set coordinate origin.

（2）The effect of G92 is to change the displayed tool nose coordinate into the set value.

（3）The X and Z following G92 can be compiled respectively, and can also be compiled wholly

2.2.45 G96—Constant linear speed cutting

Format: G96

Description: (1) The constant linear speed cutting only be suitable for system with analogue output, which controls infinitely variable speeds of spindle.

（2）When G96 is implemented, CNC regards the cutting speed of this time as the benchmark, and the spindle rotating speed will be regulated in linear according to whether it's feed or retracting in X direction.

（3）When spindle rotating speed reaches rated maximal rotating speed of system, the continuous feed is implemented in X direction, and the spindle rotating speed no longer changes.

（4） G96 is canceled by instructions such as G97, M05, M02 and so on.

（5） Because the spindle frequency converter has time setting of increase/decrease speed, so when G96 takes effect, the shorter the increase/decrease speed time constant, the better the following of spindle rotating speed.

2.2.46 G97—Cancel constant linear speed cutting

Format: G97

2.2.47 G98—Cancel feed of each rotation

Format: G98

2.2.48 G99—Setting feed of each rotation

Format: G99F××××××

Description: The F×××××× following G99 is the feed distance of each rotation, which the unit is Ųm, namely only integer can follow F. This is different from feed of every minute

2.3 Subsidiary function (M function)

The M function is also called subsidiary function, which is used for state control of CNC input and output. The subsidiary function is composed of M with two digits following it, and the subsidiary functions of numerical control system are as follows:

M00 Program pause

M01 Condition pause

M02 Program end

M03 Spindle positive rotation

M04 Spindle reverse

M05 Spindle stop

M08 Open cooling fluid

M09 Close cooling fluid

M10 Work piece clamping

M11 Work piece loosing

M12 Spindle high gear relay is on

M13 Spindle high gear relay is off

M20 Open specified relay

M21 Close specified relay

M24 Setting cutting tool compensation number

M25 Parallel tool changing time to wait for the end of the tool change

M28 Servo spindle is set for speed mode

M29 Servo spindle is set for position mode

M41～M44 Specify spindle gear rotation speed

M71 ~ M85 M function pulse output

M function is the one to switch on or switch off machine tool external switches such as spindle startup, stop and refrigeration motor switching on or switching off. M function is always different from the one prescribed by standard owing to differences of machine tool manufacturer and machine tool structure as well as model.

Then M functions will be elaborated on.

2.3.1 M00--Program pause

Format: M00

Description: M00 appears in the program, and pauses after this segment program completes. Pressing the processing start key, and the program continues.

2.3.2 M01--Condition pause

Format: M01 K×× or M01 L××

Description: The two digits after K (or L) is corresponding to certain number of I／O, and the program will stop here until the exterior inputs one low-valid (or high-valid) signal to the I／O, then program will continue. The valid time of external level is required greater than 15 milliseconds. K is high-level valid and L is low-level valid. The input port number definition of system is stored in the system, and the input number of each input port can be checked in the parameter-diagnosis interface

2.3.3 M02--Program completion

Format: M02

Description: (1) M02 completes processing program

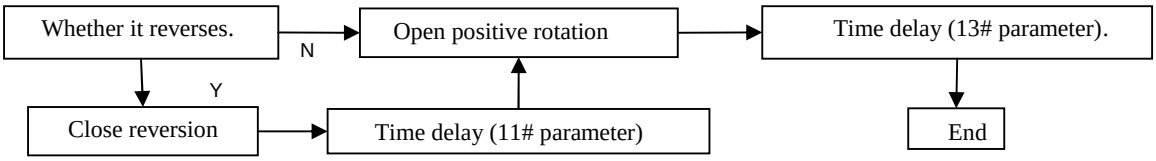
(2) When M02 isn't programmed in the program, if the digit parameter B003=0, close the spindle (M05) and cool (M09) after the program ends.

If the digit parameter B003=1 and M02 isn't programmed, not to close spindle and cool after the program ends, and only complete this cycle.

2.3.4 M03--Spindle positive rotation

Format: M03

Description: (1) The M03 instruction is in the program, first the spindle positive rotation relay will be switched on, then S function outputs analogue and control spindle to rotate in clockwise direction. It controls 1# relay. M03 movement sequence:

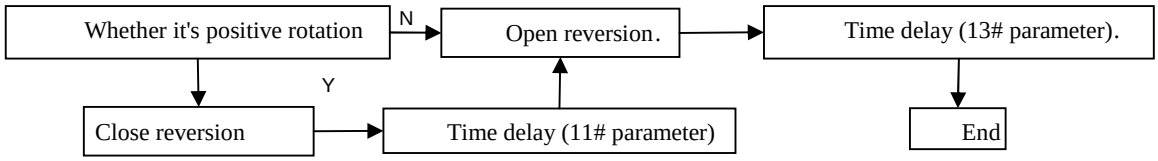


(2)If B012 is equal to 0, M03 will maintain output

(3)If B012 is equal to 1, M03 is the pulse output, and pulse delay is determined by 13# parameter

2.3.5 M04—Spindle reversion

Format: M04



Description: (1) Controlling 2# relay and starting spindle reversion.M04 movement sequence is:

(2)If B012 is equal to 0, M04 will maintain output

(3）If B012 is equal to 1, M04 is the pulse output, and pulse delay is determined by 15#

parameter

2.3.6 M05--Spindle stop

Format: M05

Description: (1) M05 instruction outputs pulse signal and closes spindle positive rotation or reversion control relay to stop outputting analogue, and the spindle rotation stops. The width of output pulse signal is determined by 14# parameter Control M05 relay power to output

(2) If P12# parameter value isn't equal to 0, the system will also output short signal to the braking relay, to provide spindle braking function.

(3) If B013 parameter is equal to 1, M05 will close S1∽S4 relays; if B013 is equal to 0, M05 won't close S1∽S4 relays.

(4)The execution course of M05 instruction is:

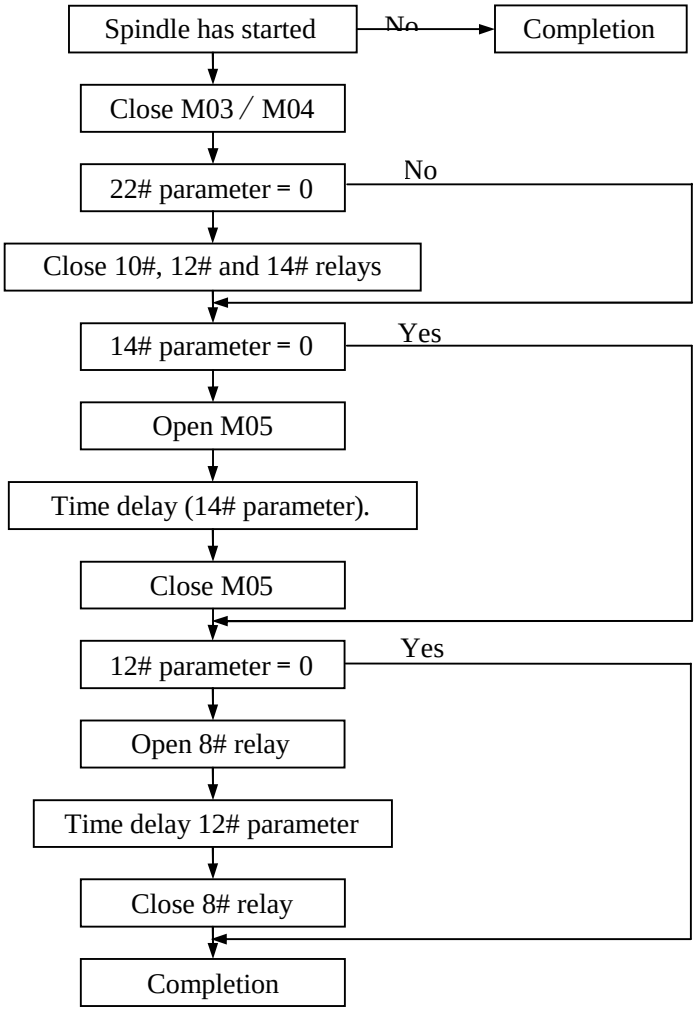


Figure 2-29 The execution course of M05 instruction

2.3.7 M08--Open cooling fluid

Format: M08

Description: M08 function will be executed at the beginning of this segment program, and the cooling fluid control relay will be switched on (M08 relay)

2.3.8 M09—Close cooling fluid

Format: M09

Description: M09 function will close cooling fluid control relay (M08 relay) after this segment program completes.

2.3.9 M10/M11--Spindle clamping and loosing control

Format: M10 or M11

Description: See description in Chapter SIX.

2.3.10 M12/M13 --Spindle high gear relay on/off

M12/M13 is used to open/close the spindle high/low gear shift relay, and the system specifies that M12/M13 relay is jointly used with S3.

2.3.11 M20—Open specified relay

Format: M20 K××

Description: The two digits after K is corresponding to certain relay number, and output of OC port will be valid when executing to this statement, which switches on the relay on the relay panel and continues after delaying time for 0.02 second

2.3.12 M21—Close specified relay

Format: M21 K××

Description: The two digits after K is corresponding to certain relay number, and output of OC port will be invalid when executing to this statement, which switches off the relay on the relay panel and continues after delaying time for 0.02 second

2.3.13 M24—Setting cutting tool compensation number

Format: M24K××

Description: When the cutting tool compensation number is used for the first time and must be changed, it only changes cutting tool compensation number instead of regulating work piece coordinate, and the two digits after K is from 00 to 10 M24 can't be used in the program.

2.3.14 M25--Wait for too-change end

When parallel tool-change is adopted, the motor of each coordinate axis will still move during tool-change course. If operator wants to start the next segment processing after tool-change is completed, M25 command can be added in program. The next segment processing shall be started after tool-change ends, to prevent tool collision. If no tool collision risk exists during retracting course, M25 command won't be required, to enhance processing efficiency

2.3.15 M28/M29-- Spindle speed/position mode:

M28: Output YTRF low level to make spindle servo enter speed mode, which is used for normal cutting.

M29: Output YTRF high level to make spindle servo enter position mode, which implements interpolation with other axes (X, Z).

See 6.3.5 in Chapter Five for details.

2.3.16 M41～M44 --Specify spindle speed gear

Suitable for the mechanical shift spindle driven by the frequency converter, see 6.3.1 in Chapter Five

2.3.17 M71~M85—M function pulse output

Format: M77（taking M77 for example）

Description: Considering that various machine tools have different requirements to M function, the system sets up this function which is used to control relay on the relay board to output short time on-off signals, and its movement sequence is as follows (taking M77 for example):

(1)Controlling 7# relay on the relay board and switching on it.

(2)Time delay 15# parameter. When 15# parameter is equal to 0, the time delay will be 0.4 second.

(3)Switching off 7# relay

2.4 F、S、T functions

F, S and T functions are the abbreviation of feed function, spindle function and cutting tool function.

2.4.1 F——Feed function

The feed function is called F function in general, and F function can directly provide feed speeds of G01, G02 and G03. F function is signified by F and following digits, which the feed speed is mm/min. The feed speed of this system is between 1 mm/min to 15,000 mm/min, and user can randomly select according to actual cutting condition. Once F is set, it can be only modified by F value of posterior statements.

2.4.2 S—Spindle rotation speed control

S0-S5 （system saves S0-S5, which are gear change signal control of multi-speed motor), the spindle control is divided into the frequency control motor and spindle-use change speed motor (two-speed motor and three-speed motor).

2.4.2.1 The spindle with frequency conversion motor

The spindle control is implemented by the S××××, M03, M04, M05, 03# system parameter, 04# system parameter and 11#～14# system parameters which determines the analogue output controlled by the spindle (default is 0-10 V). In general, the machine tool spindle has one level manual gear shift to implement various rotating speed scope, which the bigger output torque still exist at low speed. The system can output high speed and low speed analog voltage, and the top limit of high and low gear is determined by 3# and 4# system parameters in the system parameters.

Fro instance:

03# system parameter = 5000，04# system parameter =1500,

When it's at the high gear (the 'H/L input' must be switched off with the '24V earth'):

$$V_S = (V_{ref} \times S) / 03\#\,\text{system parameter} \qquad \text{(Formula 2-3)}$$

When it's at the low gear (the 'H/L input' must be switched on with the '24V earth'):

$$V_S = (V_{ref} \times S) / 04\#\,\text{system parameter} \qquad \text{(Formula 2-4)}$$

In the formula: V_{ref}: Maximal analog voltage 10V;

V_s: The analog voltage actually output by CNC;

03# parameter: High gear top limit;

04# parameter: Low gear top limit;

2.4.2.2 The spindle operation controlled by digital quantity

The instructions are S1, S2 and S3 (suitable for multi-speed motor), and used with M03, M04 and M05. M03, M04 and M05 determine the positive rotation of spindle;

2.4.3 T—Cutting tool function

The cutting tool function is also called T function used to select cutting tools, and the use of electro-holder or tool array are determined by 05# parameter (=0: electro-holder, =1tool array).The cutting tool function is signified by T and following digital block. Towards the electro-holder, the revolving holder is controlled to implement tool-change and change corresponding cutting tool number; and towards tool array, the numbers are all 0, only the cutting tool compensation number is changed

2.4.3.1 T function format

Tn.m n: cutting tool number （1—8） m: cutting tool compensation number （1—10）

There are several following written forms:

	Electro-holder	tool array
Tn	Changing No. n cutting tool and using No. n cutting tool compensation	Using No. n cutting tool compensation
Tn.0 or Tn.	Changing No. n cutting tool without cutting tool compensation	As above
T0.m or T.m	Not changing cutting tool and using No. m cutting tool compensation	Using No. m cutting tool compensation

Tn.m	Changing No. n cutting tool and using No. m cutting tool compensation	Using No. m cutting tool compensation
T0.0	No movement	No movement

2.4.3.2 The cutting tool number and cutting tool compensation number

Towards the tool array, the tool numbers are all 0, and the deviation of tool nose will be modified through changing tool number; towards the electro-holder, the tool numbers are acquired by sensor in the holder, and they are not memorized by CNC. Accordingly, no matte whether it's tool array or electro-holder, CNC can correctly learn about the tool number. However, the cutting tool compensation number can't be acquired through external switch, especially towards tool array and when the tool number is different from cutting tool compensation number, CNC can only acquire the cutting tool compensation number via memory. In normal work, the cutting tool compensation number is between 1 and 10, but under special condition, such as CNC is used for the first time, the system is totally cleared, or the cutting tool compensation number exceeds the scope above caused by disordered memory or it's not exact, the cutting tool compensation number can be specified artificially with M24, but at this time the CNC actual coordinate is probably different from displayed coordinate, and the re-centering CNC coordinate is required.

Note: If the parameter is set for electro-holder (05# parameter = 0), while CNC system isn't connected with the electro-holder or is wrongly connected, then the system will appear error 06

The physical meaning of cutting tool compensation table is the difference of tool nose among various tools. In order to decrease memory capacitance, the general cutting tool compensation table only memorizes coordinates in X and Z directions relative to one fixed benchmark of every tool, then at the time of cutting tool compensation, according to different difference of the current tool and the previous one relative to this fixed benchmark, which the offset between the two tools can be acquired, that is the cutting tool compensation table doesn't memorize the cutting tool compensation value, and the cutting tool compensation value can be acquired through calculation at the time of tool-change. 31series has 10 cutting tool parameter addresses in total, namely from T01-T10.Under PARAM status, 10 cutting tool parameters are displayed in the screen by pressing n key, and DX is the cutting tool compensation value in X direction, and DZ is the cutting tool compensation value in Z direction. R is the corner radius, PH is the phase relationship of the tool nose and the workpiece relative position

2.4.3.3 Movement sequence of electrical holder function

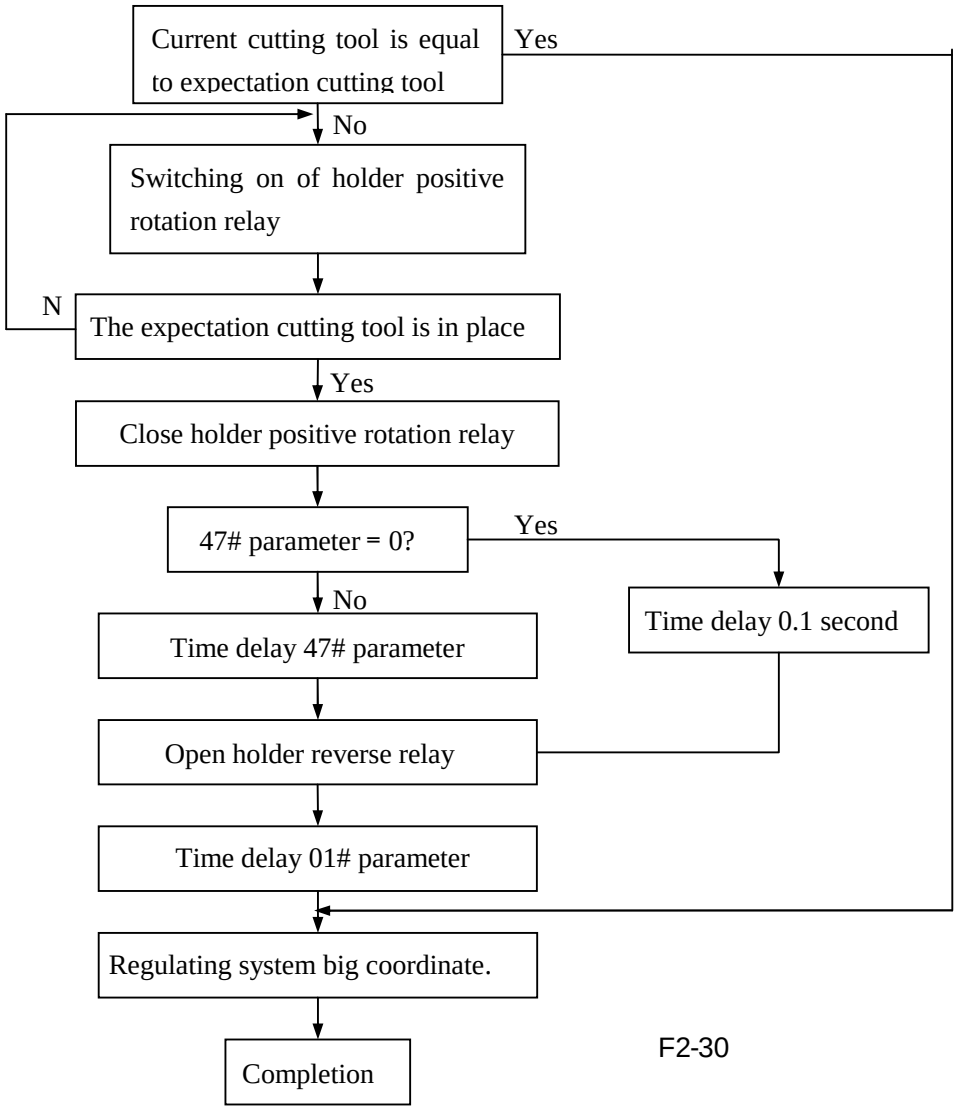
P5# of the system is used to set different tool-change modes:

P5#=0 Tool array

P5#=1 Normal electrical holder.

P5#=2∽10: Other holder defined by customer.

For the electrical holder, the movement sequence of default built-in PLC of function is as Figure 2-30:



F2-30

2.4.3.4 Presetting cutter methods and steps

This system adopts universal presetting cutter method of trial cut, and for the convenience of operation the system offers independent memory presetting cutter mode in X and Z directions, and the presetting cutter in X and Z directions is memorized simultaneously, the specific operation steps are as follows:

(1) Independent memory presetting cutter in X and Z directions

Presetting cutter method in X direction

①One work blank piece is clamped on the chuck; Accessing manual operation mode

② Rotate holder and select the cutter number requiring presetting cutter such as "T1"; then select proper spindle speed and manual feed speed and start spindle.

③ Move cutter and turn a segment of ex-circle (or inner hole) on the workblank with the prepared cutter, press "Xsav" button and "X Memory" will appear at the top right corner of the screen;

④Manually quitting cutting tool and stopping the spindle, measuring the diameter of excircle (inner bore) after cutting,

Pressing "cutting tool compensation setup" key to access the cutter compensation parameter interface, and "X 0.000" appears in the screen, inputting the diameter measured in the step 5 on the keyboard, and pressing "ENTER"

Note: If the cutting tool cuts on the other side of the work piece axial line (namely the cutting tool is in the opposite direction of axial line), then the input diameter is negative.

Presetting method in Z direction

①. Accessing manual operation mode;

②. Re-starting the spindle and moving holder, and using No. n cutting tool to turn one end surface on the work blank. "Z memory" will appear at the top right corner in the screen by pressing "Zsav" key;

③. Manually quitting the cutting tool and stopping the spindle, measuring the length L from the cutting end surface to the chuck end surface;

④. Pressing "cutting tool compensation setup" key to access the cutter compensation parameter interface, and "Z0.000" will appear in the screen. Inputting the length L measured in the step ③ on the keyboard and pressing "ENTER" for confirmation, then pressing "Save Open program" for saving. At this time, the relative position of this cutting tool in the work piece coordinate system has been confirmed.

Note: At the time of presetting cutter for every tool, the measurement benchmark must be the same; else the cutting tool compensation value in Z direction will be incorrect.

(2) Presetting cutter is memorized in X direction and Z direction

① Clamp a workblank part on the chuck, enter manual operating mode and rotate holder, and then select the cutter requiring presetting cutter such as "T1"

② Select proper spindle speed and manual feed speed, start spindle, move holder and turn an end surface by using the prepared T1 cutter on the workblank; press "Zsav" button to exit to proper position along X direction; process a segment of ex-circle and press "Xsav" button;

③ Cutter exits and spindle stops, measure the ex-circle diameter after being turned and the length L from the workpiece end surface to the chuck end surface;

④ Press **[**CUTTING TOOL**]**to enter the cutter compensation parameter interface, and "X0.000" appears on the screen, then press "ENTER" button to confirm after inputting the diameter. The cursor stops after the X-direction compensation value and flickers, press "F3 (cutter compensation)" and "Z0.000" will appear on the screen; press **[**SAVE/OPEN**]**to save after inputting the length value L. After the cutter compensation is established, the relative position of this cutter in the workpiece coordinate system will be confirmed at the same time.

Description: When the cutter is preset for the first time after clearing the memory, "F3 (Cutter compensation)" button must be pressed after entering the cutter compensation parameter interface, then the system will appear prompt of "X0.000" or "Z0.00", and the X-direction and Z-direction cutter compensation value can be input. Press "ENTER" button to confirm and then press **[**SAVE/OPEN**]**to save.

For other functions of cutter compensation such as cutter compensation modification, see cutter parameter part of operating description.

Chapter Three System Operation

Properly operating numerical control system, must master the operating methods and displayed meanings of all sorts of information of various functions. The operational interface which is offered by numerical control system to users is as follows:

1. Keyboard panel: Receiving user's instructions to system, and accordingly coordinating system internal status to implement entire system functions

2. Communication interface: The communication can be implemented with any computer equipped with standard RS 232 serial interface.

3. 6 Inch LCD, which can offer various real-time system information

4. Various input/out interface.

5. USB interface: Insert U disk to switch processing program or parameter file with the control system.

3.1 Safety protection and compensation

In general situation, the ring opening driving of stepping machine won't influence much on the machinery at the time of over travel blocking rotation owing to its principle, while towards the system which the exchange servo is as the executing component, the overload capacity of exchange servo and output torque will increase dramatically, and the mechanical damage even serious accident will probably occur. So the safety protection of machine tool is significantly important towards the machine tool driven by exchange servo-unit. The system limits probability of error occurring through various aspects below.

3.1.1 Emergency stop

The emergency button should be equipped with a pair of normally opened/normal closed contact for each, and the normally opened contact should be connected to system
(see technical manual)., so as to enter emergency stop state when the emergency button is pressed.
It's strongly recommended that the normally closed contact is switched in the strong power circuit of machine tool, which is the control circuit supplying power for main circuit (spindle and servo), so as to guaranty that the spindle and servo can stop operating with the maximal reliability under emergency.

When the emergency stop signal is received by the system, it'll be switched to the manual mode and 55# alarm occurs, and all operations are locked.

3.1.2 Hard limit

Towards the machine tool which the exchange servo is as the executing component, the mechanical type two-way or three-way travel switch should be equipped, which can cut off the power supply control circuit of main circuit forcedly when the soft limit doesn't work (see emergency stop). Towards general three-way switch: One-way is switched in the strong power control circuit, one-way is switched in the limit input of system, and the second way is as the coarse positioning signal of returning to machine tool zero.

Owing to that the action of proximity switch can't directly cut off the control circuit, so it's not recommended to use inductive type proximity switch as the limit switch, if it must be used, the proximity switch with OC port output of NPN type should be selected.

3.1.3 Soft limit

The system offers internal timing examination function to implement real-time monitoring of system coordinate whether it exceeds the interval set artificially, once it's exceeded, the operation should be stopped and switched to manual mode with 40# alarm; this course is composed of a series of reference system.

(1)Whether the machine tool coordinate (SCOR=1)(reference point) or the working coordinate (SCOR=0, big coordinate) is as the benchmark of soft limit coordinate is determined by the SCOR digit of 09# digit parameter.

(2)Whether the soft limit function is valid after returning to machine tool reference point (SNZ=0) or needn't returning to reference point (SNZ=1) is determined by the SNZ digit of 02#.

(3)The interval set by soft limit is defined by the system parameter (P parameter) 60#---65#, once the coordinate (machine tool coordinate or work piece coordinate) selected by the system exceeds interval of each axis, the system will give alarm (see parameter table).

(4)When the limit occurs, whether each axis stops suddenly or stops with speed reduction is determined by OVS of 09# towards hard limit, and determined by the SLS digit of 02# towards to soft limit.

(5)When the soft/hard limit stops with speed reduction, its time constant of negative acceleration is determined by the 44# parameter, while the maximal speeds are all set as 10 m/min.

(6)When limit sudden stop is selected, the servo will generate great impact and generate adverse effect to the machine tool, so adopting stop with speed reduction is recommended, but over-impact (过冲) will be caused and the soft limit region is exceeded; the smaller its acceleration is, the longer the interval-exceeding is, so the method of using time constant (44#) to increase acceleration and decrease border crossing length.

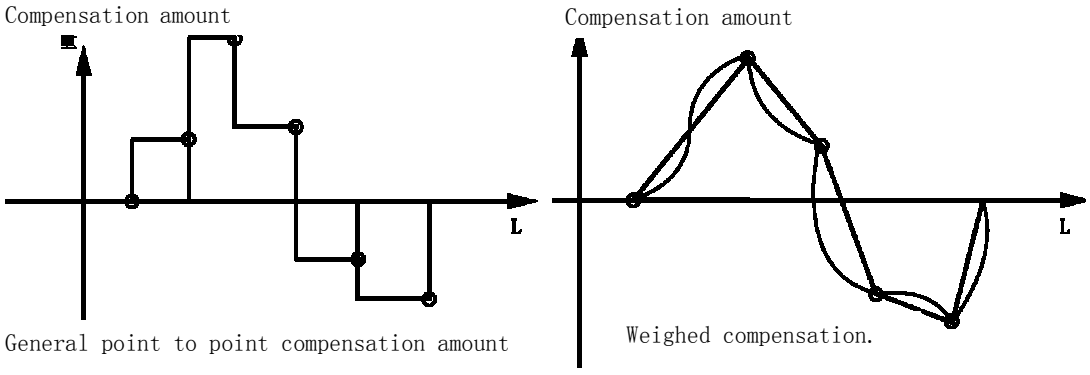
3.1.4 Clearance compensation

Towards the mechanical transmission machinery with certain reversion clearance compensation, the system can compensate the precision loss but it can't be expected that the effect after compensation is the same as the gapless processing quality, especially when the arc processing exceeds quadrant, the bigger the clearance value is, the more influence on the quality, so the machine tool should decrease the clearance compensation value as far as possible.

The system adopts principle of extra-movement to deal with clearance, and its acceleration of movement is calculated by the 39# time constant and 48# compensation speed top limit

3.1.5 Screw thread pitch compensation

Owing to influence of composite factors such as manufacture and temperature, the screw thread pitch error statistically belongs to the system error and not the accidental error, and the numerical control system thinks that when the measurement distance of error between two adjacent measuring points is small enough (≤1.5-2 times of thread pitch) , the error presents weighted linear distribution. When the system implements thread pitch error compensation, it not only guaranties the accurate compensation of the measuring point, but also the weighted compensation can be implemented to the errors among measuring points, so as to guaranty that every system cycle (about 4ms) can implement compensation to the screw error within the total travel of entire screw instead of only compensating the measuring point independently.



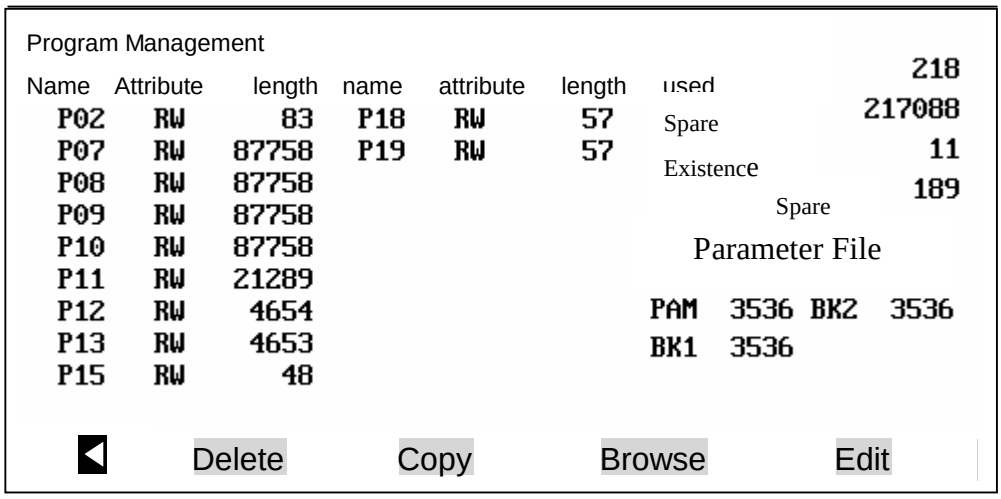
General point to point compensation amount

Weighed compensation.

Seen from the weighted compensation figure, the compensation amount of any point of the screw length L is different from the adjacent measuring point, while the compensation amount is not only related with the error value of two measuring points at both sides of current point, but also related with the fore-and-aft error change rule of adjacent measuring point. Seeing 4.5.1 for the implementation of thread pitch compensation

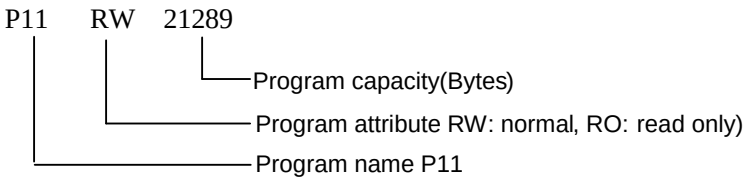
3.2 PRGRM (Program) primary function

PRGRM（Program）is the user processing program management, after pressing "PRGRM" (Program) key, the main image of "PRGRM" (Program) is displayed in the screen such as the figure below:



F3-1 Program management

User can most save 120 processing programs in the system，Towards every program, the system displays:



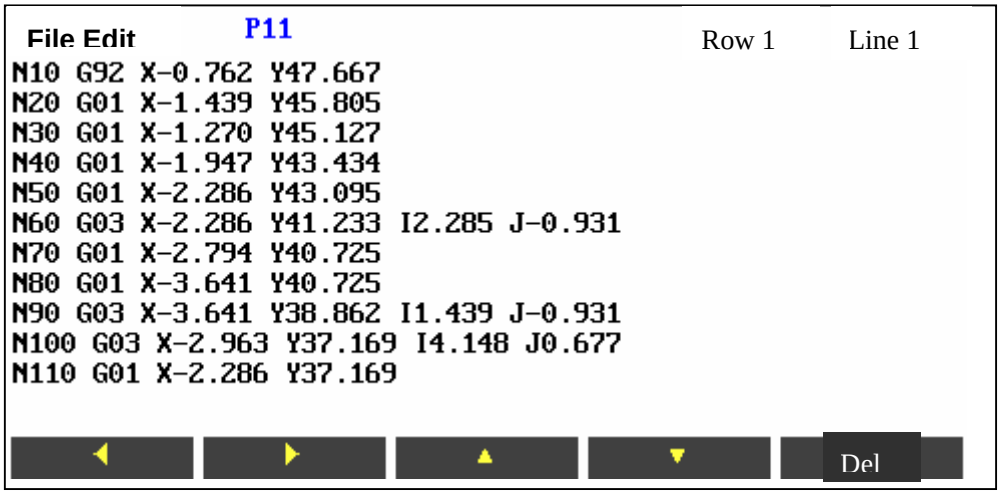
3.2.1 Input principle of program name

In the system, only the main program can process, and the main program regards P as the first letter. The sub-program regards N as the first letter and it can only called by main program. The main program or sub-program following two digits signifies various program numbers, and 8T system provides that the main program can be any one from N00 to N99, and sub-program can be any one from N00 to N99.

When inputting program name, first typing in P (or N), then inputting two digits by pressing keys from 0 to 9, if 27# parameter is equal to 0 after pressing enter key, the system won't automatically generate block number, and when 27# parameter is not equal to 0, the block number will be automatically generated. The block number increment is 27# parameter, and system will process the input program name when input completes.

3.2.2 Program edit

Pressing F5 in the program management image, which the word "edit" appears in the right of the third line, simultaneously the cursor flashes after the "Program name" in the first line. User can input one main program name P00～P99 or sub-program name N00-N99, and access edit page after pressing enter key. The edit page is as follows:



F3-2 Program edit

If this program has existed in the system, then it will be displayed in the screen. If it's the new program, there is no program in the program display region; the blank in the middle of screen is for user to edit

The current program name as well as line number and row number are displayed in the first line; X and Z are the coordinate values which are relative to program origin of two machine tool coordinate axes. The bottom line is the note of edit function, and F function key becomes cursor movement key when editing program by this time. "◄►" is used to switch cursor function key. The program display region is in the middle, and user can edit program in it

Character key and numerical key: The pressed character and number will appear in the cursor position with pressing character key or numerical key, simultaneously cursor and following character all move backward one character position, namely one character inserts in the cursor position (insert mode)

Edit function key: It's composed of F function key and enter key ENTER. The role of edit function key is to move cursor to proper position, so as to add, delete one (or one line) character and they don't directly add character in the program. The role of enter key is to move cursor to the head of next line. If it's in the certain line of program, it will insert one blank line between this line and the next one, and simultaneously bring all characters after cursor position of original line to the blank and generate one new line. " ◄► " can be used to switch functions of F1~F5 keys。

For instance: Edit the two segments of programs below:

N0010 G00 X100

N0020 G01 Z10

The sequence of key-press is：N—0—0—1—0—G—0—0—X—1—0—0—ENTER

N—0—0—2—0—G—0—1—Z—1—0—ENTER

The following keys are valid when editing：

F1: Cursor moves to the left for one character and the program content changes none. It's invalid to press F1 when cursor reaches the line head.

F2: Cursor moves to the right for one character and the program content changes none. It's invalid to press F2 when cursor reaches line end.

F3: Cursor moves upward one line, if this line is the first one when cursor reaches it of program region in the screen, it will be invalid to press F3. If it's not the first line of program, the programs in the full screen move downward one line with pressing F3.

F4: Cursor moves downward one line, if this line is the last one when cursor reaches it of program region in the screen, it will be invalid to press F4.If it's not the last one of program, then the programs in the screen move upward one line.

F5: Deleting one character before cursor, and simultaneously all the characters of this line after this character move forward one character position, so as to fill up vacant position. Pressing F5 will move this line to the end of the previous line when cursor is in the line head.

After pressing "◄►"key:

F1: Move cursor to line head

F2: Move cursor to line end

F3: Move cursor to file head

F4: Move cursor to file end

Note:

(1) If any character is not input or all the characters in the old program are deleted when editing new program, this program won't be saved which is equal to be deleted.

(2) Abnormally quitting edit mode is strictly prohibited, else part of the program even the whole program will be corrupted, these abnormal program-quitting conditions include:

A) Pressing reset key when editing, shutdown or instant power down of power grid.

B) If one large-size program is edited, the system will probably access edit page after waiting for a moment. Part of or whole program will be corrupted if the conditions above occur when waiting.

If this condition occurs, this system won't guaranty completeness of program.

If you want to quit edit mode, pressing certain primary function key and the system will be switched to other primary function status. The system will automatically save the user's edited program after processing before quitting, and the program will be recorded in the directory which is reflected in the program tab

3.2.3 Copy, delete and program status

The three sub-functions under PRGRM are principally used for changing the program integral status.

3.2.3.1 Copy (F3)

It will copy certain program to another program. Pressing "PRGRM (program)" to access program management, then pressing "F3" to display 'Copy', and simultaneously cursor appears after the first line 'program'. Typing in the copied program name in the cursor position such as P67 (P67 must be the program which has been in the system). One 'arrow' appears after this program name after pressing enter key, and cursor flashes after this arrow, then inputting the copied program name such as P68 (P68 must be the program which is not in the system). The copy completes after pressing "ENTER" and new program generates, simultaneously corresponding changes occur in the program table.

3.2.3.2 Delete (F2)

This function is to delete useless programs from the system, and they can be main program or sub-program so long as they appear in the table. Steps:

1.Pressing "PRGRM (program)" to access program management

2. Pressing "F2" and the function notes 'delete', then inputting program name which is required deleting.

3. This program is deleted after pressing "ENTER" key.

Note: Once the program is deleted, it can't be recovered, so it's necessary for you to carefully operate when deleting

3.2.3.3 Program status

Every program of numerical control system can have four attributes, which are:

A) common b) read only

Towards to common attributes, the general edit, delete etc. can be implemented. Towards to read only attribute, the system can be only displayed in the screen for viewing, which the character can't be

increased and decreased or deleted wholly. This function can prevent disoperation to corrupt program.

The hiding program can be edited, modified as the common attribute, however the program name isn't displayed in the table, accordingly no one but the operator can operate this program.

The read only hiding is the combination of two attributes of hiding program and read only.

Operating steps:

(1)Pressing "PRGRM (program)" to access program management

(2)Pressing " ◄► "

(3)Pressing "F5", and the display function notes 'program status', simultaneously inputting program name in the cursor position.

(4)The "arrow" appears post-program name after pressing "ENTER", which signifies that the program will be assigned attributes.

(5)One of four numbers 0, 1, 2 and 3 is put in the cursor position and pressing enter key, then this program name will be assigned attribute, simultaneously there are also changes in the program name table, and the meanings of four numbers are:

0: common attribute 1: read only

The default attribute is 0 (common attribute) after editing new program

3.2.4 input and output function

The system inputs and outputs program via RS—232 serial port.

3.2.4.2 Input (F3)

One program can be input from PG machine or other numerical control systems through serial port. Operation:

1.Pressing "PRGRM (program)" to access program management

2.Pressing" ◄► "

3.Pressing "F2" to display 'Input', and cursor appears and flashes after the first line 'Program', inputting program name in the cursor position (it must be the program which isn't in the system), and the input process starts after pressing enter key.

4.The input characters can be seen in turn which are displayed in the screen when communication transfers

3.2.4.3 Output（F2）

It outputs some one program to the PG machine or other numerical control systems which is stored in the system. Steps:

1 Pressing "PRGRM (program)" to access program management

2 Pressing"◄►"

3 Pressing "F3" to display 'Output', and cursor appears and flashes after the first line 'Program', inputting program name in the cursor position (it has been stored in the system), and the output process starts after pressing enter key.

4.If the character is typed wrongly when inputting program name, the "F1" key can be used for modification.

5 Towards to the specific operations which implements program exchanges with computer, please see description files in the communication floppy disk

3.2.5 List

The list function can rapidly browse and edit user program content in the CNC storage area, even if the hiding program can be displayed under this function. However, if the program is read only, then edit can't be implemented, and the operating sequence is as follows:

1.Pressing "PRGRM (program)" to access "Program management" mode

2.Pressing "F4" to access list sub-function, and displaying the first program in the CNC user program area. This program name (PXX or NXX) is displayed after 'Program' in the first line of the screen,

and the program contents are displayed in other regions of screen. Now, the function keys of F1~F5 are redefined.

F1：Edit. Pressing "F1"can edit current displayed program contents.

F2: Previous page. If current program contents are relatively more, which can't be displayed in one screen, and the "F2" key can be pressed to skip to the previous page.

F3: Next page. If current program contents are relatively more, which can't be displayed in one screen, and the "F3" key can be pressed to skip to the next page.

F4: Next segment of program. The next program name and contents of CNC user program area can be displayed by pressing "F4". If current program is the last one of the user program area, then the first program is displayed after pressing this key.

F5: Previous segment of program. The previous program name and contents of CNC user program area can be displayed by pressing "F5". If current program is the first one of the user program area, then the last program contents and name are displayed after pressing "F5"

3.3 USB disk management

The system has USB port and support access to USB disk. You can import or export the files between USB disk and user programmer storage in the USB disk management interface.

3.3.1USB disk management introduction

1. It supporting the USB disk storage adopting USB1.1/USB2.0 protocol, the file format in the USB disk is FAT format.

2. Maximum 6 levels of directories operation able in the USB disk.

3. The showing format of USB disk file is 8.3Format: ××××××××.×××Format，the file name longer than 8.3 format should be reduced to 8.3 format. It supporting Chinese directory name.

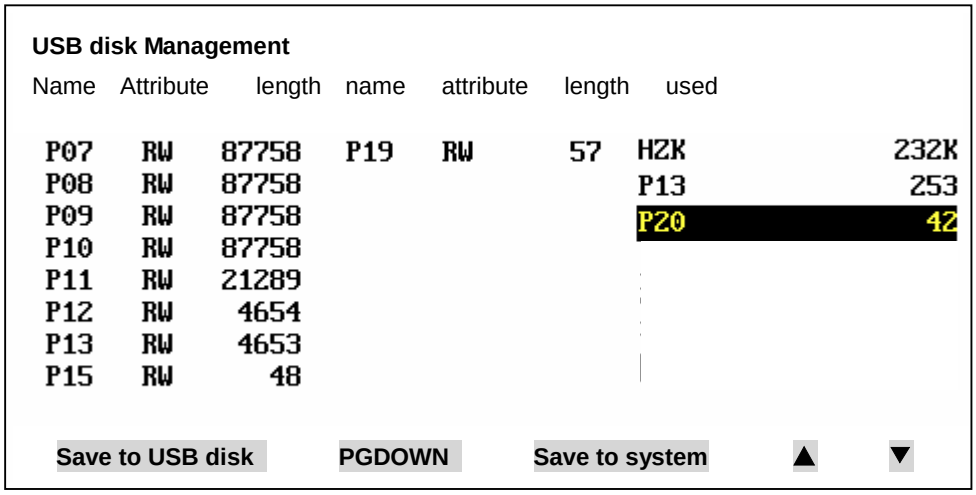
4. Supporting the mutual storage between the user programmer storage and the USB disk.

5. Automatically sorting to the USB file names.

3.3.2 How to enter USB disk management interface and interface introduction

1. Open the USB disk cap, insert USB disk to the USB port;

2. Press PRGRM, enter programmer management interface See Illustration 3-3：



F3-3 USB disk management

（Programmer list area of USB disk: showing the file list in USB disk current directory

User programmer list area: showing the programmer list in user programmer storage

Prompt box: operation prompt information display area

USB disk directory display area: showing the directory of current file in the USB disk, maximum 6 levels.

Alert box: error report for disoperation prompt box

Status box: showing the information of operation result

Introduction to F function buttons in USB disk management interface:

Save to USB disk: save the programmer in the user programmer storage to the current directory in the USB disk

Browse programmer: browse the programs in user programmer list

Circulation page up: the user program list will display 30 user programmers per page, when the programmer number exceeds on page, you may use circulation page up to show the programmers those are not displayed in the user programmer list

Delete file in USB disk: delete the selected file in the USB disk

Remove USB disk：execute this operation to pull out the USB disk

Browse USB file：browse the selected file in the USB disk

Save to system：save the selected file in the USB disk to user programmer storage

Operation buttons to display the file list in the USB disk:

PGUP ：display the prior page of current file directory in the USB disk

PGDOWN：display the next page of current file directory in the USB disk

3.3.3 Select file from USB disk

Because of Chinese character or long file name, it is inconvenience to input the file name from USB disk. To convenience operation to USB file, the system use cursor selection method to select the file for operation. The system rules: you must select the file before any operation to the file in USB disk. The selection is to move the cursor in the USB file list and locate it position on the file name and that file is to be operated, it means the file has been selected. Selection a file folder is the same operation. See Illustration5-8, the current selected file is P01.

3.3.4 How to open the file folder in USB disk

The system supporting directory depth is 6 levels in maximum, all the file folders can be opened if the directory less than directory depth. You may categories the processing programmers into the relevant file folder and it is convenience to find and management the programmers. To open a certain file folder, select the file folder and press ENTER, then open the file folder and display the content of the file folder in the USB disk file list.

If there is file folder named as “program” in the USB root directory, we illustrate the operation steps to open a file folder by opening the above file folder.

1. Move the USB cursor and select the “program” file folder;
2. Press ENTER, and the content of the file folder will be displayed in the USB file list.

3.3.5 How to return to parent directory

The following operation can return to parent directory of current file in USB disk.

1. Select[..];
2. Press ENTER, and return to parent directory, if returned successfully, it will show new path in the USB disk path display area, and prompt box shows information “return to parent directory accomplished”;
3. If the current directory is the root directory of USB disk, it can not return to upper level.

3.3.6 How to save file from USB disk to system

This function can save the file in the USB disk to user programmer storage. See the detailed operation process in below example: there is a file named “test.nc” in the USB disk root directory and to save it to the user programmer storage as the name of P03.

1. Enter USB disk management interface（See operation in Section 5.3.2）select “test.nc” file;

2. Press F8（save to system），it shows in the prompt box “please input another name for the target file” input box , input P03 and press ENTER, it will operate to save the USB disk file into the system;

3. If operation succeed, the P03 file will be displayed in the user programmer list and the prompt box shows information as “copy accomplished”.

Note: 1. The name of the file to save into the system must comply the programmer nomination rules, that is to say, it must start with letter P or N and followed by a two digits number.

2. The size of the USB file to save into the system can not exceed the maximum user programmer size the system permitted, it is 52Kbyte.

3. File folder can not be save into the system.

3.3.7 How to save file from user program to USB disk

Example: if there is a programmer named P03 in the user programmer list, to save it into the USB root directory and name as P01.

1. Enter the USB management interface（See operation in Section 5.3.2）;

2. In the USB disk management interface, press F1（save to USB disk）, it shows in the prompt box: “please input source file name” input box and cursor flickers in the input box, input P03 and press ENTER, it shows in the prompt box: “please input the target file name” input box and cursor flickers in the input box, input P01 and press ENTER，it will operate to save the P03 file from the user programmer storage to the USB disk root directory;

3. If operation succeed, the P01 file will be displayed in the USB disk file list and the prompt box shows information as “copy accomplished”.

3.3.8 How to browse files in USB drive

In the USB disk management interface, you can browse the USB disk file content directly to confirm the file is correct before USB disk file operation as save or delete. See detailed operation steps below:

1. Enter the USB management interface（See operation in Section 5.3.2）and select the file to browse in the USB disk management interface;

2. Press F7（browse USB disk file），and switch to USB disk file browse interface, see Illustration 5-9;

In the file browse interface, the following button operations are available: PGUP，PGDOWN, F4(Programmer start)，F5(programmer end)，F2(home)，F3(end)，and cursor buttons of ←,↑,→,↓, operation mode is the same as Section 5.2.6.

3. Press F8（return），exit browse interface and return to USB disk management interface. Or press any one main function buttons of RGRM, OPERT, PARA, MONITOR to exit browse function and enter corresponding main function interface;

4. If to browse next file in the USB disk, you should exit USB file browse interface(see the third step) and return to USB disk management interface, select the file to browse in the USB disk management interface and repeat the second step.

3.3.9 How to delete files in USB disk

When USB disk does not have enough remain space, you may use delete USB disk file function to delete unused file in the USB disk. Operation step below:

1. Enter USB disk management interface（ See operation in section 5.3.2），enter the directory in which is the file to be deleted（see operation in section 5.3.4），select the file to be deleted in the USB disk;

2. Press F5 (delete USB file)，it will delete the file;

3. If delete file successfully, the deleted file name will disappear from the directory, the file list in the directory will be refreshed, and prompt box shows “delete accomplished” information.

Note: 97T/98T isn't provided with the this function.

3.3.10 How to browse programmer in user programmer management storage under USB disk management interface

This function is to convenience browse check the current programmer is correct or not before you save the programmer to the USB disk. See example to browse the P03 programmer in the user programmer list, detailed operation below:

1. Press F2 (browse programmer) in the USB disk management interface, it will popup input box with “please input source file name” and cursor flickers in the input box.

2. Input P03, that is the name of the file to be browsed, press ENTER and it switches to file browse interface and display the content of P03. See the operation in file browse interface in Section 5.2.6;

3. Press F8 (exit) in the file browse interface and returns to USB disk management interface.

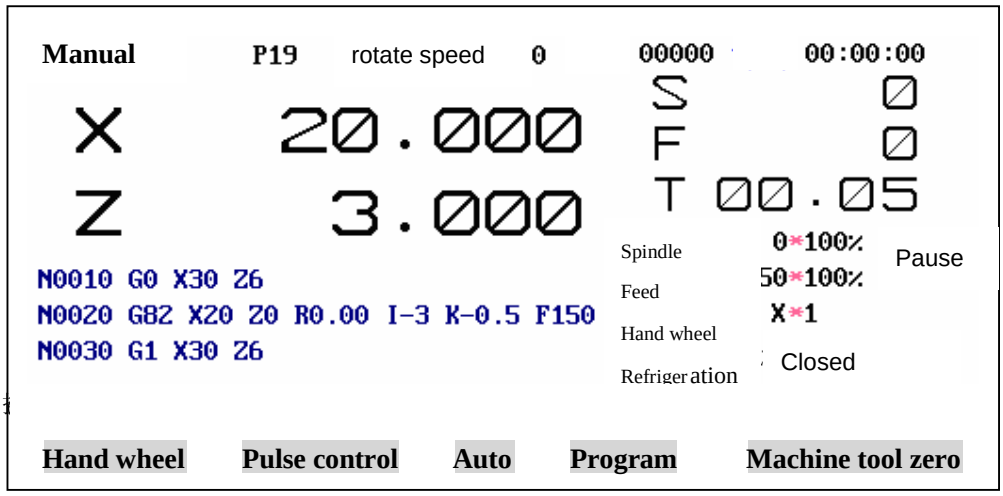
3.3.11 How to browse user programmer in circulation under USB disk management interface

When the programmer number in the user programmer list exceeds a screen(30) range, this function can browse a user programmer name if it is not listed in current screen. See detailed operation below:

1. Press F3(circulate page up) in the USB disk management interface, the programmer list box will display the programmer names of the next page, if it reaches the end page of the list, press circulate page up will show the content in the first page of programmer list.

2. Circulate page up is only valid to user programmer list in the USB disk management interface.

3.4 OPERT (Process) primary function rotate speed



Related description of operation processing interface:

(1) and Z (coordinate value displayed with big character) are the coordinate values corresponding to programming zero point, namely work piece coordinate

(2) XP and ZP are the coordinate values of cutter relative to machine tool zero point, namely machine tool coordinate, they are not displayed in the above figure, switch display mode with ALT key, and display XP and ZP.

(3) Operation mode: automatic, manual, hand-wheel, inching and machine tool zero return, displayed on the top left corner of screen. The operation mode displayed on the above figure: manual.

(4) Processing program name: program used by automatic processing.

(5) Spindle actual speed.

(6) Piece rate, time rate: display quantity of processing work piece and processing time

consumption dynamically.

(7) Programming spindle speed: spindle speed programmed by user at automatic/MDI

(8) Actual feed speed: display actual feed speed of dynamically

The XP and ZP of first line are the coordinate values which are relative to machine tool zero. The coordinate value which is displayed with big characters is the one that is relative to programming zero; various functions notes of F function key are displayed in the bottom line; and various states displayed with light background respectively indicate the operation mode, machine tool status, some parameter values of manual operation such as manual speed, spindle rotating speed and so on

3.4.1 Auto-cycle (including start processing in the position of any block number)

Auto-cycle is corresponding to automatic mode of machine tool. 'Auto' will be displayed in the operation mode window after pressing "F3" key, then pressing "Save Open program" key. The cursor appears and flashes after 'Program' of the first line in the screen and user can input program name which is ready for operation in this position. "F1" is used to modify wrong-typed characters when inputting program name; the 'program name table' in the middle of screen display existent program name, program size and program attributes in the user program area

The program name input is right and pressing "ENTER", which describes that system has prepared to execute the program such as pressing "Cycle start", and the program starts operation. When pressing "F2 (line number)", the cursor appears after the program name. It notes user to input line number which is the line to execute line skip. The program before line number is not executed.

This system provides that the programming zero under auto-cycle is the point which the value of system big coordinate is zero, and all programming tracks regard this coordinate as the benchmark. The programming coordinate system is repeated with the one of work piece

For instance：The programming is：

G90 G01 X10 Z30 F100

However, when cycle starts, the big coordinate of system is：

X —50.000

Z 3.500

Then the results of the execution above are: X coordinate moves 30 mm along the positive direction, and Y coordinate moves 26.5 mm along the negative direction, in addition, Z coordinate moves 26.5 mm along positive direction.

After pressing "Cycle start" key, the system firstly implements necessary internal processing such as examination, error detection etc. If error occurs, the note of error will appear; otherwise the program will be executed in order.。

Note: (1) The program name of auto-cycle processing will be called out by file open key.

(2) The figure simulation or track will be accessed under automatic processing, and see 3.5 section for specific contents

3.4.2 Manual operation of machine tool

Manual operation includes two modes of manual continuous feed and stepping feed, which are called manual mode and pulse control mode, and both refrigeration and spindle can be manually operated.

(1) Manual mode: Manual mode can be accessed by pressing "PRGRM (Program)" key, and under the manual mode, ◄Z-、Z+►、X-▲、X+▼ signify operating keys which are used for various coordinate axes to move along their positive directions or negative directions. When pressing one of them, the corresponding coordinate axis will move long its direction. The feed speed can be manually set by pressing "FSET". When pressing one of feed keys above and "  " simultaneously, the speed

operation can be set by pressing parameter.

(2)Pulse control mode: It's same as the manual mode that presses "F2" key to access pulse control operating mode. The coordinate moves some given length along its corresponding direction by every coordinate feed key pressing. This length is set by "ISET".

(3)Settings for manual operation parameter: They are only valid in manual and pulse control modes.

Pressing "FSET" key: Settings for coordinate movement speed in manual or pulse control modes (letter key region "FSET" key), and the cursor will appear after word 'Feed' with pressing "FSET", then the digits can be input, which signify millimeters for feed per minute. It'll be valid after pressing "ENTER".

The scope of this speed is 1～6000mm／min（when it's 0.001mm pulse equivalent), if input has error, the system will automatically set for 50.00mm ／min."F1" key can be used to modify the wrongly-typed digits.

Pressing "I set" key: Setting up pulse control stepping amount ("I set" key in the letter key region), it's only valid in the pulse control which is used to set up stepping amount.

Pressing "I set" and inputting stepping amount after cursor (0.001～65.5mm).

Pressing "S set" key: Setting up speed of spindle ("Sset" key in letter key region), and inputting digits in the cursor position which signifies speed of spindle. It'll valid after pressing "ENTER". The system outputs the analogue which is correspondent to rotating speed. The top limit of this rotating speed is set with 3# parameter and 4# parameter.

(4)Spindle and refrigeration: In the modes of manual, hand wheel and pulse control, both spindle and refrigeration can be manually operated

3.4.3 Return to machine tool zero

Under OPERT (Process) function, pressing "F5", the operating mode will become returning to machine tool machine tool zero mode, and three words of 'machine tool zero' will appear after 'Operation'. "Cycle start" will be pressed and the selected axis will return to zero in the positive direction after pressing "F4" to type in the zero-returning axes X and Z. After pressing ◄Z-、Z+►、X-▲、X+▼ the system will return to zero according to the principle of “X” axis first then “Z” axis. In the continual repeated processing, in order to eliminate accumulated error, it is recommended that zero-returning be implemented once at intervals. It is recommended that zero should also be returned once after starting up each time, which makes system have one absolute reference benchmark, simultaneously eliminates position shift of various coordinate axes for shutdown last time

3.4.4 Hand wheel (hand-operated impulse generator)

Towards to lathe control system, hand wheel can control machine tool's rectilinear movement in directions of X or Z. Hand wheel sets up three speed gears (multiply factors) for various requirement which are respectively X1, X10 and X100. Switch can be implements among speed gears, and the minimum control precision is identical with system control one, while the maximal control speed is 100 times than the pulse equivalent. Hand wheel is principally designed for fast rectilinear movement, presetting cutter etc. of machine tool.

Hand wheel operation is as follows:

(1)Pressing "OPERT" primary function key in the main menu to access machine tool operating interface.

(2)Pressing "F1" to access hand wheel operating mode.

(3)Pressing "F4" key to select movement coordinate axis, simultaneously the selected coordinate axes are displayed under 'Operation' prompt in the screen.

(4)Pressing "  " key to select hand wheel multiply factor, simultaneously the selected

multiply factor is displayed under 'Operation' prompt in the screen.

(5)Turning hand wheel, and the machine tool makes response movement.

(6)To quit hand wheel status, either "F" function key (F1—F5) can be pressed

3.4.5 System status setup

Under 'Auto' sub-function of "OPERT (Process)", there are two statuses setting up related with machine tool-numerical control system.

(1)Single segment: Pressing "F1" key for toggle. When it's valid, 'single segment' is displayed after 'Operation'. The system will stop to wait for user to input with executing block of each time. Pressing "Cycle start" key for each, the system will execute one block downwards. If the "Cycle cancel" key is pressed when waiting, this cycle will be cancelled and can't be recovered.

(2) Pause: It's correspondent to "Cycle start" that auto-cycle can be paused and processed by "Cycle pause". Pressing "Cycle start" after pause to continue, if pressing "Cycle cancel", this automatic processing will be cancelled.

Note: The track display function can be accessed after automatic processing starts.

3.4.6 MDI operating mode

In the manual, auto-pulse control and hand wheel modes of "OPERT (Process)" primary function, the cursor appears in the second line of the screen with pressing "GMDI", now one line program can be typed in and the system will be executed after pressing "ENTER" key. This segment of program requires no block number. In the automatic mode, all executable blocks can be executed in MDI mode

3.5 Figure display function

Under "OPERT (Process)" primary function, the automatic mode can be accessed by pressing "F3 (Auto)", and now pressing "F5 (track display)" to access figure display mode (online or simulation).This function is used to display movement track of tool nose under certain processing program control. This function can make user directly view the operational process of programming track, and at the same time, big errors in the program can be evidently found combining coordinate values which are displayed in the screen. The figure in the screen can track tool nose's cutting movement in synchronization, and displays appearance of bar. The operator can directly view the manufacture course of revolving body from the screen when tool nose moves. In the simulated condition, the cutting tool center track is displayed in the screen, however various coordinate axes of machine tool don't move, in addition, control functions of various machine tool electrical equipments are invalid which are mainly for debugging program. When program has no errors, it can be processing to avoid failures even accidents owing to programming oversight.

3.5.1 Image access sequence of figure display function

1.Pressing "OPERT（Process）" key

2.Pressing “F3（Auto）” key

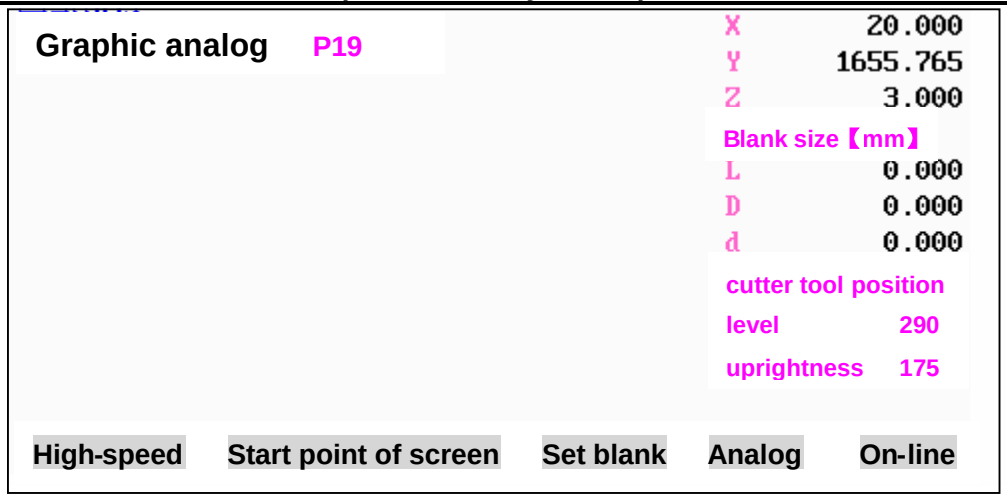
3.Pressing “Save Open program” key

4.Inputting file name “PXX”

5.Pressing “ENTER”

6.Pressing “F5（track display）”

Screen displays as follows:



F3-5 Graphic analog

Function of buttons in graphic display interface:

1 Low-speed movement: change the moving speed of analog cutter, after this button is pressed, the button becomes fast movement. This button is switched between the low-speed movement and fast movement according to status.

2. Start point of screen: move analog cutter to zero position On-line

3. Set blank: used to set blank size. Press this key and then move cursor to set or modify the blank size. Input the actual blank length after L, input the blank outer diameter after D, and input blank inner diameter after D.

4. Analog: press F key to realize graphic analog, the actual cutter won't move.

5. On-line: press this key to realize graphic analog, and actual cutter also moves.

After 【◄►】 key is pressed, 【◄】, 【►】, 【▲】 and 【▼】 keys are displayed on the F function key, to move analog cutter.

3.5.2 Graphic analog procedure

1. In the graphic display interface, press the F function key corresponding to "Set blank".

2. Input the blank length after L in turn, input the blank outer diameter after D, and input the blank inner hole after D (if there is no inner hole, input 0).

3. There is one vertical line in the right lower part of figure display region, and upper end of it signifies the cutting tool center. Because the system doesn't learn about movement direction and course of track, so user must move small cross line to select one proper position in the screen where it stops, which is the work piece coordinate point currently displayed.

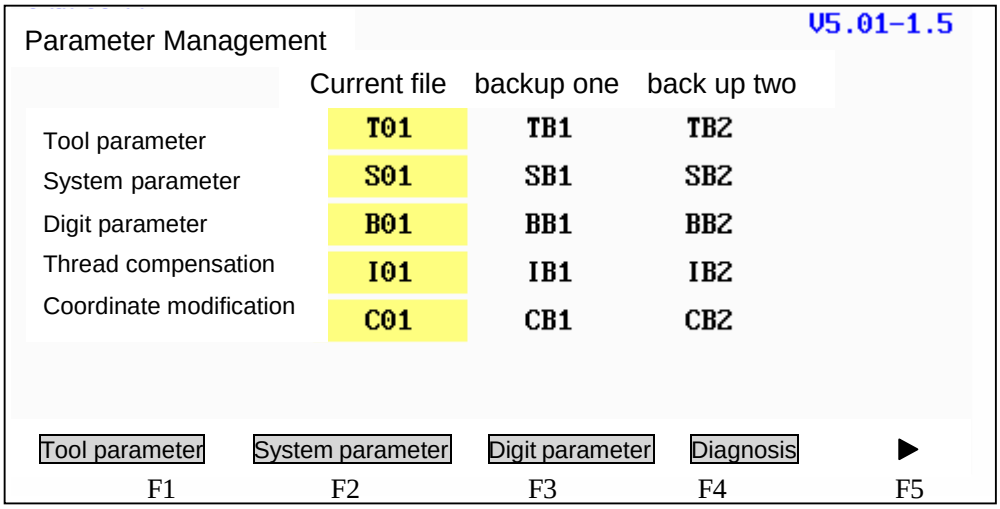
F1, F2, F3 and F4 can be used to move vertical, and the movement direction is the arrow one which is correspondent to F1—F4 in the screen. While pressing F function key every time, the movement amount of vertical can be one pixel/five pixels. How to select movement amount is determined by F5. Two digits of 1 and 5 alternately appear in the second region of the screen with pressing "F5". Therefore F5 can move the vertical center to the optional position of figure display area through F1-F4.

Once vertical is in place, the program will start execution with pressing "Cycle start" key. The track starts from the upper end of vertical, and the programming track is simulated according to program requirements. However, whether the machine tool operates depends on drive power and simulation selection or online. When program execution completes, other primary function keys can be pressed to quit figure track display. After simulation completes, whether the program is right can be judged through figure. If there is error, pressing "PRGRM (Program)" primary function key to return to edit mode for program modification. When the program modification completes, the figure simulation will be implemented again until it's right. After simulation completes, the figure track display can only be quitted

Chapter four system function

4.1 parameter system

PARAM (Parameter) primary functions are the parameter setting and status display of machine tool-numerical control system. As one of numerical control systems, there are some parameters which should be set up by users. Under the PARAM (Parameter) primary function, the available sub-functions can also be selected by F1～F5 keys, and the PARAM (Parameter) image accessed is as follows:



F4-1 Parameter system

Parameters can be divided into three categories below:

1 Examination category: Testing whether the external input port status, spindle rotating speed and spindle encoder operate normally.

2 System setup category: Initializing system, setting up password and formatting electronic disk and so on.

3 Parameter system:

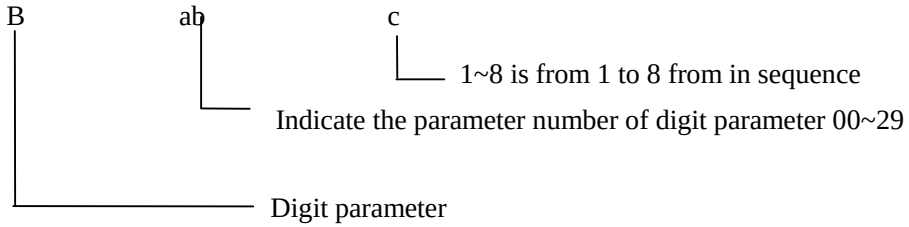
(1) System parameter: (P parameter) 00#—B9#(p199#)

(2) Cutting tool compensation parameter: 24cutting tool numbers

(3) Thread pitch error compensation: 160 points/axis, three axes of X, Y and Z in total.

(4) Digit parameter: 8-bit parameters of 40 and 320 statuses

Description: There are 120 system parameters from P00# to P119#, and the system parameter will be represented by PA0# from P100#, and P110# is indicated by PBO#; there are 30 digit parameters by analogy, and each digit parameter has 8 bits, and each bit is indicated by Babc as follows:



For instance: B124: the fourth bit from the left of the 12# digit parameter; lubricate and open in fixed time

Because the various parameter files are stored in the electronic disk via different modes, the system divides the parameter files into three types: current parameter, backup I and backup II, and their key distinction is as follows:

1. The current parameter is being used by the numerical control system, its filename is displayed in the parameter display area of program management interface. It's stored in the user program storage area, the format of electronic disk will delete current pramater file, however, format won't influence backup I and backup II of parameter file
2. The user can browse and edit current parameter, and the backup parameter can be only browsed.

Before the user browses or edits the parameter file, the user shall confirm whether the "Parameter storage type" selects current parameter or the parameter file of backup I and backup II, the backup file can't be edited, when the backup file is edited, the status bar will prompt "This file belongs to backup which can't be edited". The backup file is used to backup current parameter file, when it's necessary to edit or use certain one backup file, it shall be recovered to current file and then it can be edited or used.

The user can use cursor 【F5】(►) to move the parameter storage class cursor, to select the storage class of parameter file which will be opened

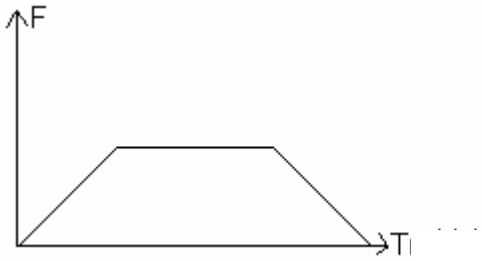
4.2 Basic conception of parameter

Before introducing parameter, some basic conceptions should be introduced first

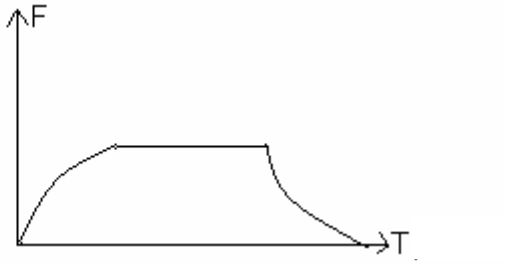
4.2.1 Time constant of speed increase and decrease

It's the time that the operating speed consumes from zero to maximum set one, or from maximum set speed to stop towards to the coordinate axis of machine tool.

When the system positions thread cutting rapidly at G00 and performs manual feed, it'll adopt straight line speed increase and decrease, when machining is performed and hand-operated pulse generator is adopted, the exponent speed increase and decrease will be adopted.

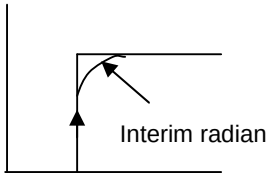


Straight line speed increase and decrease



Exponent speed increase and decrease

The interim radian (as Figure) will occur between the cutting block by adopting the exponential type speed increase and decrease, this radian is related with the feed speed and exponent speed increase and decrease time constant (39#), it's favorable to reduce this radian through reducing 39#, if the interim radian isn't generated between two sections of traces, the fast angle clear command G61 and G62 can be adopted (refer to G61 and G62 description for details).



4.2.2 Acceleration

numerical control system adopts sampling control method and implements sampling control for one time to various coordinate axes. The speed increase and decrease of various axes adopt control method of even acceleration in every other 4.096 millisecond, namely the change of motor speed (if it operates) of each sampling interval (4.096ms) is constant value, which is the speed increase and decrease of the system. Its calculating formula is:

$$a = \frac{V_m \times Ts^2}{60 \times T_m}$$ （Formula 4-1）

Among it: Ts: System sampling time (4.096 ms)

Tm: Time constant of speed increase and decrease (ms)

Vm: Maximal speed (mm/min)

a: Acceleration

he acceleration can be changed through changing maximum speed or changing speed increase and decrease.

Generally, the maximum speed is used to limit the one in this movement status, and the acceleration value can be determined with time constant of speed increase and decrease, while the actual operating speed of system should be less than or equal to the maximum speed.

The relationship among acceleration a, maximum speed Vm and time constant Tm: "—" signifies constant, while h and Vm change at the same time, it's specifically determined by the formula：

V_m	↑	↓	–	–
T_m	-	-	↓	↑
a	↑	↓	↑	↓

4.2.3 Electronic gear ratio

Regulating system output which makes the coordinate operating value of system concord with the distance of table movement, and it's not necessary to regulate transmitting ratio of motor and screw for this.

The electronic gear is set by system parameters. Two parameters of each axis can be called multiply factor MLT and percentage DVT, and when the bearing ball screw transmitting working table operates:

$$\frac{MLT}{DVT} = \frac{Pmt \times Gf}{Pcn \times Gd}$$ (Formula 4-2)

Among it: Pmt: Pulse number of each rotation for motor, and the stepping motor: Pulse of each rotation

Servo motor: Encoder line number X encoder multiple frequency number

Pcn： Screw thread pitch μm

Gf: Total of driven wheel teeth number in the transmitting between motor and screw connection

Gd: Total of driving wheel teeth number in the transmitting between motor and screw connection

Towards to direct connection，Gf＝Gd＝1

MLT and DVT must take the smallest positive integral value.

The scope of MLT and DVT is 0—65535, but the ratio must within 0.01~100.

4.2.4 Parameter password

For preventing parameter modification without intention, the system sets up the password. First the password unlocking must be input, and then the parameter modification can be made, which the method is as follows：

1. Pressing "Cutting tool modification and regulation", and the cursor appears after "Password".
2. Inputting password which concords with the password in the "System zero setting" function.

3. Pressing "ENTER". If input is right, the system will clear away this line else display error

Note: Refer to Appendix 2 for specific definition

4.3 Digit Parameter

The digit parameter can be used for setting towards the one which has only two selective conditions and branches. Each parameter has 8 digits and each digit has only two states of 0 or 1, which can be used for switch selection of certain status, and the system has 40 digit parameters in total which determines 320 states

4.3.1 Access

Key-press sequence “PARAM （Parameter）”→“◄►”→ “F3 （digit parameter）”

The digit parameter is from 0# to 29#, each digit parameter has 8 bits, which is called first bit, second bit to eighth bit from left to right.； Each digit parameter is indicated by Babc, for instance, B134 indicates the fourth bit of No. 13 parameter

Note: Refer to Appendix 3 for specific definition

4.4 Thread pitch error compensation

Each axis of the system can be input most 150 error compensation points, and the compensation point number of each axis and interval between two points are determined by P parameter 52#—57#. The point system exceeding compensation scope regards the thread pitch error as 0, and the system between two compensation points regards the thread pitch error as linear change.

1. Access: Pressing "PARAM (Parameter)", then pressing "F4"

2. Pressing "F1 —F4" to move cursor can select error value of some one point, while the coordinate value of this point which is relative to machine tool coordinate (machine tool reference point) is in the lower part of the screen, and it's signified by XP, YP and ZP.

3. Pressing "ALT" key can skip to the next page;

4. Pressing "F5" can select axes of X, Y and Z

4.4.1 The required caution problems of thread pitch error compensation

1. The password must be input and then modification can be implemented.

2. The input value is point measurement error, namely the compensation value which must be input to offset this error.

3. Whether thread pitch errors of various axes implement compensate is determined by 01# digit parameters Zpce, Ypce and Xpce.

4. The dynamic compensation condition can set the REDP digit of 01# digit parameter as 1, and the compensation value in the operating process of various axes can be dynamically displayed on the operating interface.

5. 21DM system must first return to machine tool zero, then the thread pitch compensation can be implemented.

6. After returning to machine tool zero, the small coordinate in the top right corner is displayed as: XP 00000.000 YP 00000.00 ZP00000.000 require compensation, and XP, ZP coordinates must move to the negative direction.

7. The thread pitch error compensation interval length and compensation point number of X direction and Z direction can be determined according to screw length, screw precision and processing work piece size of them. Every axis most compensates 150 points (52, 53, 56 and 57parameters).

8. Using laser interferometer to check errors of every other distance (52, 56 parameters) along negative direction from machine tool zero in the three directions X and Z.

9. Inputting the measured errors into the thread pitch error item in the parameter table.

10. Setting the digit parameter 03# as: XXXXX101(open reference point returning function, and X signifies that this digit can be I / 0)

4.4.2 Thread pitch error compensation example

Example: The effective length of X screw rod is 300mm, which totally compensates 100 points. 52# parameter＝300 / 100＝3, 53# parameter＝100 (≤150)

Utilizing laser interference to measure the thread pitch error:(returning to zero first which makes XP ＝0)

Moving to -3mm position along -X direction (system display), and moving to -2.974 with actual measurement, which 1# thread pitch error is -0.006

Moving to -6mm position along -X direction (system display), and moving to -6.003 with actual measurement, which 2# thread pitch error is +0.003

Moving to -9mm position along -X direction (system display), and moving to -9.007 with actual measurement, which 3# thread pitch error is +0.007

Moving to -12mm position along -X direction (system display), and moving to -11.990 with actual measurement, which 4# thread pitch error is -0.010

Moving to -15mm position along -X direction (system display), and moving to -14.998 with actual measurement, which 5# thread pitch error is -0.002

Moving to -18mm position along -X direction (system display), and moving to -17.991 with actual measurement, which 6# thread pitch error is -0.009

Moving to -21mm position along -X direction (system display), and moving to -21.001 with actual measurement, which 7# thread pitch error is +0.001

Moving to -24mm position along -X direction (system display), and moving to -24.002 with actual measurement, which 8# thread pitch error is +0.002

Moving to -27mm position along -X direction (system display), and moving to -27.009 with actual measurement, which 9# thread pitch error is +0.009

Moving to -291mm position along -X direction (system display), and moving to -291.011 with actual measurement, which 97# thread pitch error is +0.011

Moving to -294mm position along -X direction (system display), and moving to -294.000 with actual measurement, which 98# thread pitch error is 0

Moving to -297mm position along -X direction (system display), and moving to -296.999 with actual measurement, which 99# thread pitch error is -0.001

Moving to -300mm position along -X direction (system display), and moving to -300.007 with actual measurement, which 100# thread pitch error is +0.007

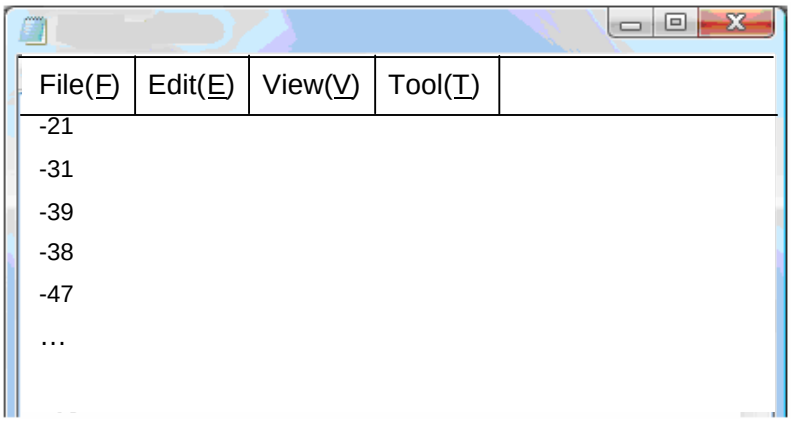
Finally saving the data and CNC system will automatically implement thread pitch compensation when processing

4.4.3 Pitch error compensation U disk import

The pitch error data measured by the laser interferometer can be input to the computer generally, and then it can be imported to system with U disk after the data is processed according to the format required by system. Most of the error data from the instrument is the integer number taking micrometer as unit, meanwhile, the requirement is as follows in the principle of convenience:

1. The pitch error data file is generated by Windows notepad, the filename of pitch error data per shaft is different: X-axis belongs "I01X.TXT", Y-axis is "I01Y.TXT", Z-axis is "I01Z.TXT", A-axis is "I01A.TXT";

The content of pitch error data file is as Figure below:



2. In the pitch error data file, the data of each line is corresponding to a parameter of pitch compensation parameter of system, the data is increased progressively, the data of each line is an integer number with symbol; as the above Figure 21, after the data is imported to system, the corresponding 1# parameter is 0.021

Operation steps:

1. In the pitch compensation interface, press 【F1】 to switch the interface to the one which shall be imported such as Y-axis;

2. Name the processed data file according the naming rule, and then copy it to the root directory of U disk;

3. Press the F function key corresponding to "U disk import" (if F function key isn't provided, press 【◄►】 to switch F function).

4.5 Cutting tool parameters

The cutter parameter of each cutter include four items which are X-direction cutter compensation value, Z-direction cutter compensation, corner radius and phase number (1～9) of tool nose direction to work piece.

About the effect of cutting tool parameter in the programming, please see 1.2.6.

(1)Pressing "F1 key" to access cutting tool parameter setting menu, and at this time the cursor flashes in the parameter position of 'T01' cutting tool number.

(2)At this time, the F function key is redefined, and pressing "F1" or "F2" keys to move cursor to the required cutting tool number position.

(3)If the X direction value is to be input, pressing "F3 (cutting tool compensation)" then directly inputting work piece diameter of trial cut, finally pressing "ENTER" to generate the cutting tool compensation.

(4)If the Z direction value is to be input, first pressing "ENTER" and pressing "F3 (cutting tool compensation)"; then inputting work piece length of trial cut, finally pressing "ENTER" to generate the cutting tool compensation.

(5)Pressing "Save Open program" key to save the input or modified parameters.

4.6 Initialization

The following functions are realized in the initialization interface: clear memory, format, modify password, recover default parameter, backup and recovery of parameter file, time set and serial number and so on.

Procedure of entering initialization interface:

1. Press main function key 【parameter】 to enter parameter interface;

2. Press 【◄►】 key and press 【F1】(initialization) to enter the initialization interface.

4.6.1Clear memory

If the abnormal condition such as system disorder, display disorder and data disorder due to due to external interference during the system operation, the memory shall be cleared. It'll set all the storage units for 0, including operation parameter of power-off protective zone and the program name table. Accordingly, the user shall implement this function to prevent the important parameter and program being lost.

Procedure of memory clear:

1. Enter the initialization interface and then press 【F1】(clear memory), and enter the memory clear interface

2. Input correct password and then press 【ENTER】

3. The system is reset automatically.

4.6.2 Format

When the user program has error, file or file directory is disordered and parameter file can't be saved, the electronic disk shall be formatted. Format will delete all user's program in the system and parameters except the backup parameter (time and password won't be influenced), user shall implement this function with care, to prevent important program and current parameter file being lost. Before format is started, current parameter file shall be saved, refer to the parameter backup operation.

Format steps:

1. Enter initialization interface and then press 【F2】(format), and enter format interface

2. Input correct password and then press 【ENTER】

3. System starts format.

4.6.3 Modify password

System adopts password to prevent some important parameter files being damaged due to incorrect operation and causing system failure. The default system password is "**XZ0012**", user can modify password in the system password modify interface.

Steps of modifying password:

1. Enter initialization interface and then press 【F3】(modify password), and enter password modify interface

2. Input the correct password in the input box of "Old password"

3. Input new password in the input box of "New password" and "Input password again", the passwords input two times shall be same

4. After password is input, press 【F4】(confirm) (or the cursor is in the input box of confirming password, and 【ENTER】 can be also pressed) to save new password.

5. When the password is modified, system will indicate "Password has been modified, please keep new password well" in the status bar, if system indicates "Password is error", please input correct password in the old password input box, else password can't be modified, if system indicates "New password isn't same as the above one, please input again", it'll mean the two passwords input in the "New password" and "Confirm password" are different, please confirm the password and then input again

4.6.4 Default value

The default value interface primarily completes the following functions: recover the default value of parameter, backup parameter to backup data zone, recover parameter from backup data zone, import from U disk and export from U disk

Enter the default value interface:

1. Press main function key 【parameter】 to enter parameter interface
2. Press 【F4】(initialization) to enter initialization interface
3. Press 【F4】(default value) to enter default value interface

4.6.4.1 How to recover the default value of parameter

The function of recovering default value of parameter is to recover the parameter of appointed parameter system to the default parameter, this function is primarily used when the system can't operate normally due to numerical control system parameter disorder, the parameter default value can ensure normal operation of system, but it won't conform to specific demands of user, accordingly, user shall implement this function with care. In order to prevent incorrect operation, password shall be input before operation

Steps of recovering default value:

1. Enter the default value interface
2. Press 【Password】 key, input correct password and then press 【ENTER】
3. Use cursor key 【►】 and 【ENTER】 to move the large cursor to the column of "Current parameter" and the line for recovery of default value of parameter
4. Press 【F1】(default value) and system starts default value operation

4.6.4.2 How to backup parameter

The main function of parameter backup is to prevent the parameter data being lost, the parameter backup file isn't impacted by format of electronic disk, accordingly, user shall keep a backup of parameter meeting its own operation parameter in the numerical control system, accordingly, the parameter can be recovered at any time when it has error. The numerical control system provides two storage spaces for each parameter, namely "Backup I" and "Backup II"

Steps of parameter backup:

1. Steps of parameter backup:
2. Press 【Password】 key, input correct password and then press 【ENTER】
3. Use cursor key 【►】 and 【ENTER】 key to move the large cursor to the column of "Backup I" or "Backup II" as well as the line for parameter backup
4. Press 【F2】(backup) and system starts backup operation

4.6.4.3 How to recover backup parameter to current parameter file

The parameter recovery is to recover the data backup by user previously to current parameter for use of numerical control system, if user doesn't backup data, this operation can be omitted. The operation of parameter recovery will overwrite current parameter, if this parameter isn't backup, it can't be recovered, accordingly, user shall implement this function with care. Before the parameter is recovered, the user shall browse the backup data which will be recovered to current parameter

Steps of recovering parameter:

1. Enter the default value interface
2. Press 【Password】 key, input correct password and then press 【ENTER】
3. Use cursor key 【►】 and 【ENTER】 key to move the large cursor to the column of "Backup I" or "Backup II" as well as the line for parameter recovery
4. Press 【F3】(recover) and system start recover operation

4.6.4.4 How to export current parameter to U disk

The current parameter of numerical control system is stored in the electronic disk in the form of file, including the following files: cutter parameter, system parameter, bit parameter, pitch compensation and coordinate bias, and the filename is respectively T01, S01, B01, I01 and C01.The export parameter is to export the current selected parameter of numerical control system to root directory of U disk, the filename

is the one in numerical control system, the "Export file" operation is only for current parameter. User can export one set of parameter meeting requirement to U disk, and then save it in computer, the parameter can be recovered when system parameter is disordered

Steps of exporting file:

1. Enter the default value interface
2. Press 【◄►】 key to switch the functions of F function key
3. Press 【Password】 key, input correct password and then press 【ENTER】
4. Use cursor key 【►】 and 【ENTER】 to move large cursor to the column of "Current parameter" as well as the line for parameter export
5. Press 【F2】(exported from U disk), after operation is completed, the corresponding parameter file will be copied to the root directory of U disk, if operation fails, the error will be prompted

4.6.4.5 How to import parameter file from U disk

The U disk import parameter file operation is to find the file whose filename is same as the large cursor under root directory of U disk, after it's found, and use the file in the U disk to replace corresponding current parameter. The "Import file" operation can be only used for current parameter, and this operation can't be used for Backup I and Backup II of numerical control system. The user shall ensure the imported parameter is correct, else it'll cause disorder parameter of numerical control system. Before these imported parameters are processed, please confirm whether the parameter is correct, else it'll damage cutter, machine-tool, work piece and injure staff

Steps of importing file:

1. Enter the default value interface
2. Press 【◄►】 key to switch the functions of F function key
3. Press 【Password】 key, input correct password and then press 【ENTER】
4. Use cursor key 【►】 and 【ENTER】 to move large cursor to the column of "Current parameter" as well as the line for parameter import
5. Press 【F4】(imported from U disk), after operation is completed, the corresponding root directory file of U disk will be copied to the system as current parameter to be used, if operation fails, the error will be prompted

4.7 Time set

In the time set interface, the user can view and set current date and time, and the data isn't impacted by the format, if the system belongs to demo version, it won't modify current time but it can be browsed

The method and procedure of time set:

1. Enter initialization interface, press 【F5】(time set), and enter time set interface
2. Move large cursor to the required position, press number key to input directly, please confirm whether the input data is correct, else the data can't be modified
3. After data is modified, press 【Save】 key to save the modified data

4.8 Coordinate modification and regulation

The holder or tool carriage of lathe will generate micro-displacement owing various causes that the work piece coordinate of tool nose will generate error, and this error can be amended by using the modification function of coordinate

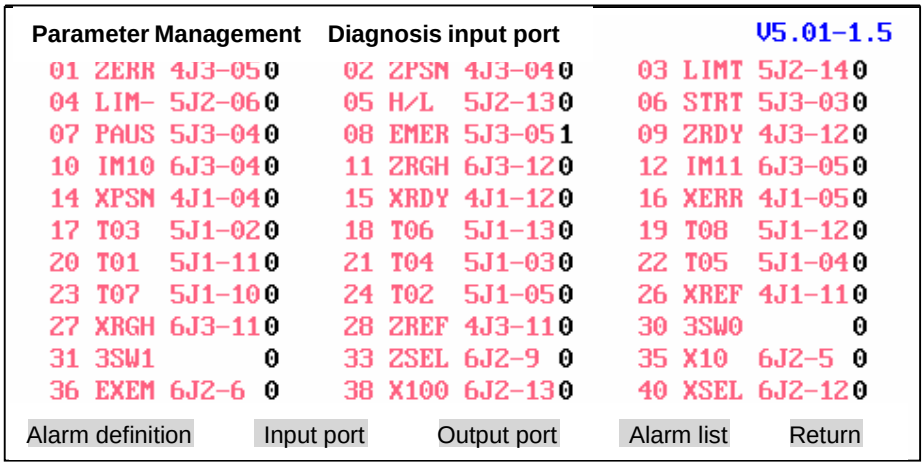
After accessing "PARAM (Parameter)" primary function, pressing "◄►" key and "F4 (modification and regulation)" key, then coordinate modification and regulation interface appears

4.9 Diagnosis (external input signal monitoring)

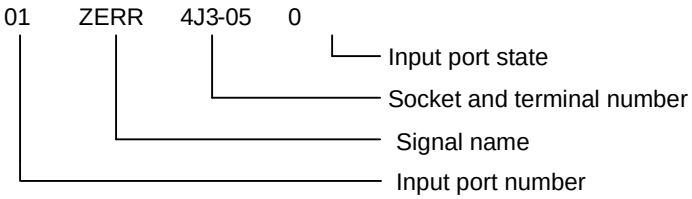
After accessing "PARAM (Parameter)" primary function, the "◄►" key should be pressed then the "F4 (Diagnosis)" key. This function will display status of external input port, such as whether there is signal input into the system, or whether external limit, zero signal are valid and so on. When certain digit displays 0, no signal is input, while it becomes 1 which signifies there is signal input of corresponding input port. As is the figure below:

4.9.1 Input port

In the diagnosis interface, press 【F2】(input port) to enter the input port interface:



Each line includes three groups of signals and each signal includes the following information (take the first group of signal as example)

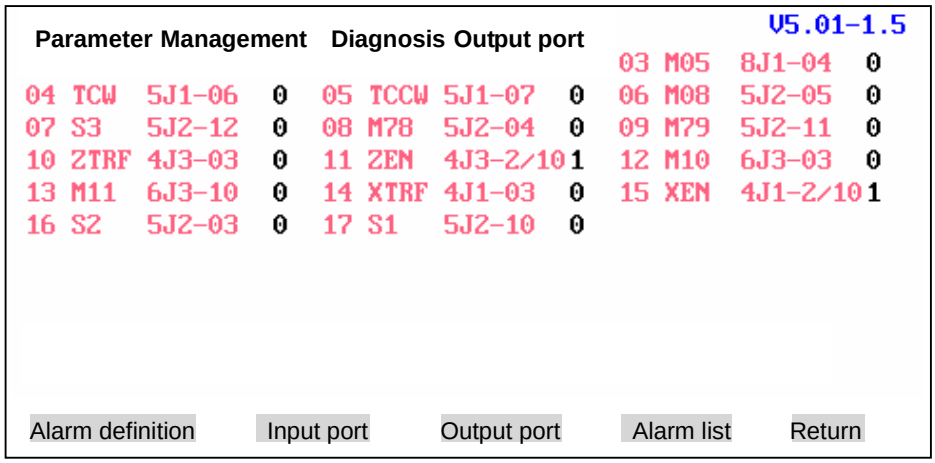


The input port number is corresponding to the input signal input interface. The input port status is corresponding to the input status of signal input interface, and it's unrelated with the active level defined by the bit parameter B04, B05, B06, B07 and B28.

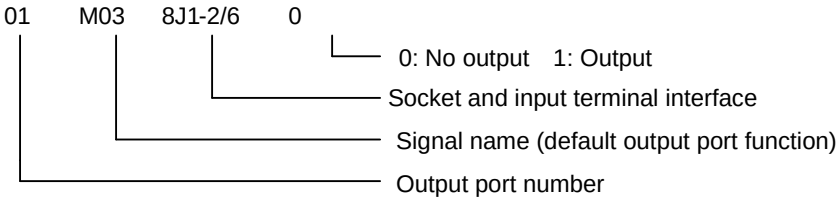
The input port of some functions can be set, for instance: the coarse positioning signal of X-axis reference point, the input port number is defined in parameter P66, and user can set it as required. If the 5J1-12 is to be set for the coarse positioning input port of X-axis, find the input port corresponding to 5J1-12 is 19, and set the parameter P66 for 19.

4.9.2 Output port

In the diagnosis interface, press 【F3】(output port) to enter output port interface:



Each line displays 3 groups of information, each group of signal includes the following information (take the first group for instance)



The output port status is the one from system to corresponding output terminal, when test is made, the signal can be output via M20KXX and M21KXX and check whether it's normal via measurement of corresponding terminal.

For some output port functions which can be set, the output port set method is same as the input port set method.

4.9.3 Spindle speed and spindle encoder

This interface primarily checks whether the spindle encoder is normal and test whether the spindle rotation speed is stable. When the spindle rotates, this function is used, the numerical control system displays the spindle speed and pulse number*4 sent by the encoder per rotation, the normal 1200 line encoder is 4800, the deviation of 3-5 pulse can be permitted, once the number is more at one time, it must be reduced in the previous/next time, and there is no accumulated error. If the data displayed for the first time to third time isn't very correct, it'll be normal

4.9.4 Hand-operated pulse generator encoder

The feedback data of hand-operated pulse generator read by system when the hand-operated pulse generator is rotated is displayed in the box of hand-operated pulse generator encoder, to test whether the hand-operated pulse generator or encoder interface is normal.

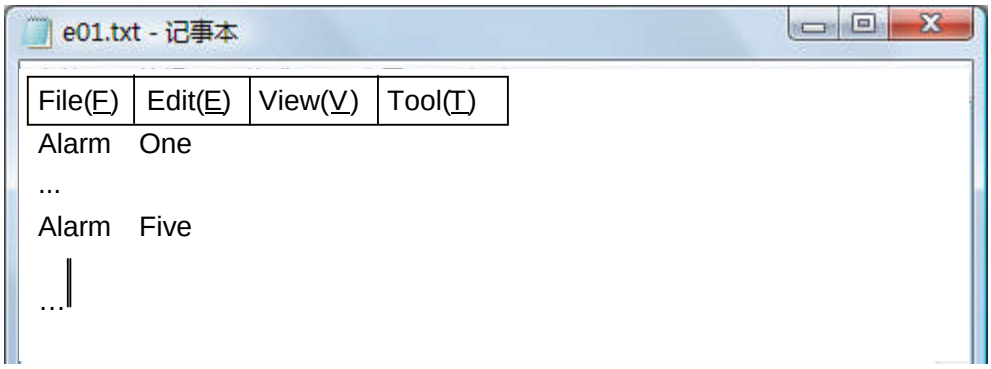
4.9.5 Alarm definition

In the diagnosis interface, press 【F1】(alarm definition) to enter the alarm definition interface. The alarm definition function provides customized alarm function, and user can define 5 customized alarm maximally, error is indicated as X1～X5 to distinguish general alarm. The customized alarm can be displayed after the system receives input port signal and determine whether to stop machining according to the parameter alarm definition parameter.

4.9.5.1 Set procedure:

(1) Edit customized alarm content file

The first 5 lines of alarm file are respectively corresponding to the alarm contents of 1#～5# customized alarm, each customized alarm system can display 15 Chinese characters maximally as Figure below:



F4-2

After the file is edited, save it, the filename is "e01.txt" and copy this file to root directory of U disk

(2) Import the alarm file to system

Enter the parameter interface according to the parameter, and then press F1 (alarm definition) to enter the customized alarm interface, press F1 (import file) to import the alarm file (e01.txt) to the system

(3) Configure relevant parameters

In the customized alarm interface, the alarm contents are included, and the parameter also includes alarm, cancel, stop and output

The role is respectively as follows:

Alarm: alarm input port, when the corresponding input port inputs an active level, system will generate a corresponding customized alarm, when it's 0, this customized alarm isn't given.

Cancel: cancel this customized alarm input port, when the cancel alarm input port to input an active level, this customized alarm will be canceled, when it's 0, user shall use CAN key to cancel alarm.

Stop: whether to stop automatic processing, when it's 0, processing won't be stopped

Output: the output port of outputting signal externally when the customized alarm is given, when it's 0, it won't output signal externally

The parameter P119 is the number of customized alarm, the maximum value is 5, when it's 0, the customized alarm function will be closed. The active level of input port can be configured via parameters B04, B05, B06, B07 and B28.

4.9.5.2 Use instance

Set the 1# customized alarm for "Alarm 1" and generate 1# customized alarm when 5J1-12 terminal inputs low level, stop machining, when the 5J1-10 terminal inputs low level simultaneously, system will clear this customized alarm

(1) Edit customized alarm content file, generate "e01.txt" file and save it to root directory of U disk, and then import the alarm contents

(2) The 5J1-12 input port is 19 and 5J1-10 input port is 23 in the diagnosis table. Move large cursor to the line with serial number of 01, input 19 in the "Alarm" column, input 23 in the "Cancel" column, input in the "Stop" column and input in the "Output" column

(3) Set the parameter P119# for 1, set the parameter B066 for 0 and set B062 for 0

4.9.6 Alarm list

In the diagnosis interface, press F4 (alarm list) to enter the alarm list interface, the error of current system can be displayed, and the alarm list is arranged according to the time when error is generated, and maximal 10 errors can be displayed simultaneously

Chapter Five System Important Functions

5.1 How to enhance processing efficiency

The methods of enhancing processing efficiency primarily include two types:

(1) Shorten or cancel acceleration and deceleration course between blocks

(2) Perform tool-change and spindle start & stop simultaneously during movement of coordinate axis.

5.1.1 Unless process requires sharp corner between two traces of work piece, don't use G61 and G62 command as possible. If the time constant of processing is 100ms, each processing program will save 0.6～0.8s.

5.1.2 Run S and T command in parallel

System provides three parallel outer commands:

(1) Motor holder tool-change

(2) Spindle start and stop

(3) Spindle clamps and loosens

Realize after the digit corresponding to 20# digit parameter (set for 1) is opened

For instance: (both digit parameter B206 parallel spindle start & stop and B208 parallel tool-change are set for 1)

```
N10 M03 S1000 T2
N20 G0 X150 Z450
N30 G01 X50 Z120 F1200 M25
N40 Z20
N50 G0 X100
N60 T1
N70 G0X150 Z450
```

Description: After N10 M03 S1000 T2 is run, system will perform tool-change of M03 S1000 and T2. Later (after about 0.008s), system will perform retracting action of G0 X150 Z450, during the course of moving to X150 Z450, system will start spindle and tool-change. About 3～6s can be saved in general.

For M25 command of N30, after system confirms tool-change in position, G01 X50 Z120 F1200 in N30 will be performed, else wait for end of tool-change, to prevent cutting tool being collided when approaching work piece, if cutting tool collision can be avoided when performing N30, M25 command can be not used.

5.2 Modify cutting tool compensation value during processing

If the work piece dimension is changed due to cutting-tool wear during processing, it can be modified and regulated timely, to reduce rejection of work piece:

(1) Execute pause or single segment.

(2) Manually stop spindle (or use three-position switch)

(3) Measure work piece dimension.

(4) Enter cutting tool compensation modification interface, and modify cutting tool compensation value according to dimension change direction.

Description:

(1) System doesn't compensate the block executed currently, but compensates it in the next segment.

(2) The modified cutting tool compensation can be both current cut and other cut.

(3) For the cutting tool compensation value modified during processing, system will modify the value instantly and re-adjust tool nose position with speed of F500 which conforms to actual size.

(4) Wrong modified value will probably cause cutting tool collision.

5.3 Spindle control

5.3.1 Spindle analog quantity output control

1. 0～5V or 0～10V can be available for system analog quantity output

B113=0：0～10V B113=1：0～5V

The spindle speed is implemented by S function, S0～S5000

S0 is spindle stop.S1～S4 are the relay output controlling multi-speed motor of spindle, S5 is to keep.

S6～S5000 (confirm the top speed of spindle according to P3# and P4#) is 255 gear analog quantity output.

2. Spindle constant linear speed output

G96: Set constant linear speed function available

G95: Cancel constant linear speed function

G96 must be used with S function, when G96 occurs, the linear speed will be calculated with the G96 spindle speed and current X coordinate; when the X coordinate is changed, the spindle analog quantity output will be changed correspondingly to maintain constant linear speed cutting, and the upper and lower limits of constant linear speed of spindle are defined by P26# and P28#.

When G96 is programmed, the work piece coordinate of X-axis can't be zero; else 53# alarm will appear owing to that the calculated linear speed is zero.

3. Output scaling of spindle analog quantity

When the spindle adopts the combination of mechanical step shift and variable frequency control (for instance, the spindle adopts double speed motor or mechanical shift), the system will be provided with following methods of automatically or manually controlling analog quantity to output, so that the spindle speed conforms to the programming speed.

(1) External low speed signal input: H/L

When B081 is equal to 0, the system will check the level of spindle high and low H/L signal to determine the spindle speed (the input number of external high and low input signal H/L is determined by P76#); when the input of input number defined by P76# is invalid (high level), the top speed of spindle will be confirmed by P3#, and when S is programmed to P3#, the analog quantity output of spindle will be maximal. When the input defined by P76# is valid (low level), the spindle speed of analog quantity output maximal voltage will be determined by P4#. When 081 is equal to 1, the system won't check H/L signal, and the parameter regarded as spindle speed when the analog quantity output is maximal will be determined according to the memorized S1～S4 states.

Corresponding spindle top speed determined by S1-S4 defined by program when analog quantity output is maximal The corresponding parameter can be specified with M41～M44 as rotation speed of maximal analog quantity (see the table above).

	S1	S2	S3	S4
Upper limit of rotation speed at all levels	P3#	P4#	P77#	P78#
	M41	M42	M43	M44

Two gears selection is automatic when B081 is equal to 0, but the user must provide external on-off signal to system. Four gears can be selected when B081 is equal to 1, and the user must use S1～S4 (or M41～M44) in the program to specify the parameter number with spindle speed, to ensure that the programmed spindle speed conforms to the real rotation speed

5.3.2 M function control of spindle

The M functions are composed of M03, M04, M05 and S function, and various control requirements can be implemented through different options of P parameter and digit parameter.

1. Various option matching of M function

B012: =0：M03 and M04 are normal state holding output which is stopped by S0 or M05.

B012：=1： M03 and M04 are pulse output, and the pulse width is determinedbyP13#.

M05 can be only pulse output and the pulse width is determined by P14#.

When M03 and M04 are switched directly, time delay will be determined by P11#.

If the spindle is required for fast dynamic braking, the delay of relay output will be determined by P12#, and the braking movement defined by P12# will occur after M03 and M04 are switched off.

The parameters can be set for 0 to make this function invalid

2. Association of spindle and M function

For convenience of programming operation, the system will define several associated options controlled by spindle

(1) B131: determine whether S function automatically executes M03 (=0 Execute, =1 Not execute)

（2）S1～S4:

B102：=0: S1～S4 are available =1: S1～S4 are not available

B132：=0: S4 is invalid, =1： S4 is valid

B133：=0: S4 is output by M78. =1: S4 is output by M79（B132=1）

B134：=0: M05 is to close analog quantity =1: M05 isn't to close analog quantity

Note: M02 command will close analog quantity and cooling fluid compulsorily (M05 and M09 will be executed compulsorily in despite of B134), if the spindle needs continual rotation after the program ends, don't use M02 command at the end of program.

5.3.3 Spindle clamping chuck (hydraulic chuck) control

1. The hydraulic chuck control modes include the following:

(1) The program command controls M10 spindle to clamp and M11 spindle is loosened.

(2) External pedal switch: including single switch, ganged switch and two-way knob-operated control.

(3) Keyboard button control: the button is switching mode (press once to clamp and press once to loosen).

2. Relevant parameter options of control mode

P95#: Relay output port number, relay which controls clamping hydraulic solenoid valve.

P96#: Relay output port number, relay which controls loosening hydraulic solenoid valve.

P97#: define external input port number and start clamping operation.

P98#: define external input port number and start loosening operation.

Pa5#: control the power-on time of hydraulic clamping solenoid valve to prevent damage of hydraulic ram owing to power-on for long time.

Pa6#: control the power-on time of hydraulic loosening solenoid valve to prevent damage of hydraulic ram owing to power-on for long time.

P105# =0 or P106# =0 are the power-on mode of oil cylinder for long time.

Pa7#: spindle clamping in-position input port number, Pa7#=0, not to check clamping in-position.

B181:=0: the clamping/loosening function isn't allowed when spindle rotates=1: clamping/loosening is allowed when spindle rotates.

B182:=0: spindle is clamped when single pedal switch is powered on, and spindle is loosened when single pedal switch is powered off.

=1: spindle is clamped when single pedal switch is stepped once to switch on, and the spindle will be loosened when it's switched on again.

B184:=0: adopt single pedal switch and one contact.

=1: adopt ganged pedal switch and two sub-contacts, one contact is stepped down and spindle is clamped (short signal).The other sub-contact is stepped down and spindle will be loosened (short signal).

When two-position knob (a pair of normally open contact) is adopt to control, B182=0 and B183=0.

5.3.4 Spindle starting state detection function

System sets two kinds of input judgment, to check whether the spindle confirms start-up after being started.

1. B141=0: check whether input is valid by external I/O port (P94# parameter definition) to determine whether the spindle has been started.

2. B141=1: when the spindle speed is checked greater than 50RPM, the system will thinks that the spindle has been started.

3. B144=0: the system won't check the spindle start-up ready signal when the spindle is started every time; =1: check whether start-up of spindle is ready.

When checking whether the spindle is started, the system will scan the input port defined by P94# in cycle within the time determined by P11#, and it'll be normal by checking that the signal is valid, else alarm will be given.

For B141=0 and B144=1, regard the normally open contact of ready output of the spindle frequency converter as the ready signal input of spindle, to avoid the failure that the system can't start the spindle normally owing to starting up delay of frequency converter.

5.3.5 Spindle position/speed mode

For some special purpose lathes, the spindle adopts spindle servo-motor, which can implement cutting processing and can control position that the spindle is as rotary shaft, so that the spindle servo operates in the speed mode (revolving body cutting) and position mode (interpolation of spindle and X-axis and Z-axis), and Y-axis of system can control spindle to operate with the two modes. The steps are as follows:

1. Connect the pulse output of system Y-axis to the pulse feeding interface of spindle servo

2. Connect the spindle analog quantity output to the analog quantity interface of spindle servo, and order ±10V analog quantity output system.

3. Connect the YTRF signal of Y-axis to the on-off input switching speed/position mode of spindle servo.

4. M28 makes YTRF low output so that the spindle servo operates in speed mode, and M29 makes YTRF high output which is in position mode.

5. When the revolving body is cut, execute M28 first which makes spindle servo enter speed mode, and cut ex-circle and end surface and so on.

6. Before setting spindle to enter position mode, execute M29 first, the Y-axis returns to its mechanical zero and implement absolute positioning, then two-axis or three-axis interpolation of Y-axis and Z-axis and X-axis can be implemented.

7. Related parameters:

B145：=0: spindle operates in normal M03，M04, analog quantity control mode.

=1: speed and position mode.

B143：=0: spindle stop （M05） command doesn't send M29.

=1: spindle stop （M05） command sends M29（enter position mode）

5.4 External function control

5.4.1 Three-position switch

The three-position switch can implement feeding under automatic processing and pause of spindle.

Note: When the three-position switch pauses or spindle stops, the spindle operation on system sub-panel is valid; once the three-position switch is recovered to program execution, the system will recover the rotation speed and rotation direction set in the automatic processing program.

Screw thread pause: only when Z-axis returns to the starting point of screw thread cycle, it'll pause and the spindle won't stop rotation.

B152:=1: Three-position switch is valid, and the start-up of any auto-cycle must turn the three-position switch to operating position

Pa2#: input terminal number executed by three-position switch program

Pa3#: input terminal number paused by three-position switch spindle

5.4.2 Control from system to feeding shaft

1. The pulse output mode of system to feed is output with feed pulse and direction pulse.

2. Primary options controlled by feeding shaft

When the following parameters are set for 1, the corresponding functions of Z-axis, Y-axis and X-axis will be respectively opened; when they are set for 1, the functions won't be opened.

1．Electronic gear:B006～B008

2．Thread pitch compensation:B016～B018

3. Reverse clearance compensation: B026～B0284. Mechanical zero: B036～B0385. Reverse operation of each axis: B086～B0886. Circle coordinate display: B106～B1087. Internal contracting brake control: B196～B198Description:

1. When the internal contracting brake is valid, the internal contracting brake of each axis is output by XTRF, YTRF and ZTRF.2. After power is on, the system must delay some time to output internal contracting brake signal after the servo ready signal is detected (time is defined by Pa4#)3. Once the system detects the driving alarm, it'll cancel all internal contracting brakes.4. The circle coordinate regards 360 degree as upper limit of coordinate display value.

5.4.3 Servo-unit and system response logic

When the power is on, strict precedence relationship is for the power-on course of servo and power-on course of system

Principle:

1. The system must be powered on first or the system and driver are powered on simultaneously

2. In general, the servo strong power supply is composed of two sets, one set is r, t which is

single-phase 220V used by the control circuit in the servo; the other set is R, S, T which is tri-phase 220V, which supplies power to AC servo-motor through large power module reversibility control, called as main circuit.

(3) For places with high requirements, it's recommended that R, S, T is powered on after R, T is powered on.

The power-on course control of system to servo is composed of the steps below.

Power-on course of servo-unit R, S, T:

1. The power-on of system is delayed, and the time is determined by PB5#; after this time, the system will think that the servo control power r, t has been powered on.2. After PB5# delay is completed, the system will output R, S, T closed M function output; the output port number of this M function port is defined by P99#. This output port can drive intermediate relay to drive AC contactor which loads R, S, T to servo driving.3. The sequence of servo output ready signal and each axis enabling signal of system output is different, and the system is determined by B084.

(1) B084=0: system will check whether the ready signal of each axis is valid, them output enabling of each axis.

(2) B084=1: system outputs enabling of each axis first, then check the ready signal of each axis.

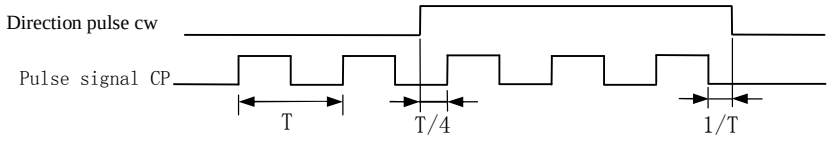
4. B031=0: system won't output enabling in any case.

B031=1: according to B084 state, system outputs enabling signal of each axis (not available)

5. B096, B097, B098=1: check ready signal of Z-axis, Y-axis and X-axis; =0: not check.Once the system check that the ready signal of each axis is valid and output enabling, meanwhile, there is no other severe alarm, the system will be in the awaiting state and can operate normally.

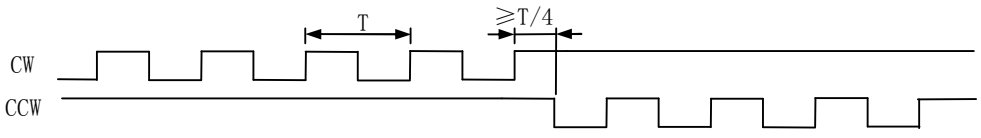
5.4.4 Pulse output mode of system to feeding shaft:

Maximal pulse frequency: 1MHz (feeding speed corresponding to 1 micrometer resolution ratio is 60m/min), pulse output wave shape：



1. The system guarantees that the front and back of direction pulse cover the pulse signal when direction is changed. One fourths pulse period2. The pulse signal is half duty ratio.3. The pulse fluctuation ratio of CP signal is equal to or less than 125ns (digit parameter B101=0) under low vibration mode, and the speed accuracy is 7.5mm/min. The average speed fluctuation is less than 0.5% under the high precision speed control mode (digit parameter B101=1).4. When big thread pitch is processed with high rotation speed, B091=1 (open feeding flexible treatment) can be set to improve thread processing quality:5. Double pulse mode (software version V5.0 above is valid, only for the numerical control system with USB interface)This mode is optional, the standard configuration isn't provided, the user can visit company website through Internet and download system software after asking for order to the company, then update the system software (or confirm in order) through USB interface.

Wave shape with double pulse mode：



5.4.5 Soft limit

Soft limit is controlled by system, and an alarm signal will be given when the work piece coordinate or machine tool coordinate moves and exceeds certain one scope and movement stops, switched to manual mode

Soft limit includes the following parameter modes

Digit parameter:

1. B021 =1：Open soft limit function

2. B024=0: soft limit can be valid after returning to mechanical zero. =1: soft limit is always valid (not recommended)3. B029=0: regard machine tool coordinate as soft limit basis =1: regard work piece coordinate as soft limit basis P parameter：

P60#～P65# are respectively limit coordinate scope of soft limit of coordinate system zero positive and negative direction selected by X-axis, Y-axis and Z-axis (see manual for parameter)

Description:

Once each axis exceeds soft limit scope when moving, the system will immediately reduce speed and stop moving, certain distance will be exceed from the soft limit space, and this exceeding distance is related with the speed when soft limit occurs; moreover, this exceeding distance is related with speed and acceleration.

5.4.6 Mechanical zero on-off setting

The mechanical zero is also called machine tool zero or reference point, which is used to recover work piece coordinate system at fixed position on the machine tool when power is on and unify important data system such as internal cutter, coordinate and protection.

In line with the operating habit of previous numerical control system, except the mode that zero must be returned in power-on (B033=0), the system also opens the mode of power-on without returning zero (B033=1).

The signal access of machine tool zero includes two modes

1. One approach switch: each axis adopts an approach switch as zero signal, which is single signal zero returning for short.2. Initial positioning switch and servo-motor Z signal zero finding mode, which is double signal for short. System recommends the mode of returning zero in power-on and the second zero switch access mode.

Operating mode of zero returning

1. Zero returning in power-on After the "Machine tool zero" mode is selected, manually operate direction feeding key, this axis finds zero, B19-4=1, it's not necessary to press direction feeding key continually.

2. Program zero returningG74: under the mode of returning zero in power-on, G74 is invalid after power-on, zero can be returned only by manually operating direction key

Parameter related with zero returning

Set basic parameter (digit parameter)

B036～B038=1: Open zero returning of relevant axis

B034=0/1: determine whether to recover work piece coordinate system

B033=0: processing can be implemented by returning zero in power-on; =1: processing can be implemented without zero returning after power-on.

B116～B118=0: Select double signal zero returning of this axis; =1: select single signal zero returning of this axis.

B136～B138=0: Find zero signal along positive direction of each axis.=1: find zero signal along negative direction of each axis.

B186～B188=0: When double signal returns zero, this axis will reversely move and finds motor Z pulse signal.

=1: when double signal returns zero, this axis won't reversely move and finds motor Z pulse signal.

5.4.7 Tool-change course

The system can handle several modes of tool array, electric holder, hydraulic/servo driven holder, and the user defines the tool-change mode by P5# parameter:

P5#	0	1	2	3	4	5	6	7	8	9	10
	Tool array	Electric holder	2～10 are defined by user (option)								

10 tool-change forms can be defined, the standard configuration of system is 0 and 1, and other holder control user must order in advance or describe holder form in advance.

The system can control 10 cutters maximally

P2# defines cutter quantity

Parameters related with electric holder and tool-change is as follows:

P1#: reversing time when holder is clamped. Too long time will cause overheating of holder motor, too short time will cause that the holder won't be clamped.

P2#: cutter quantity on electric holder, P2#=4, T05～T08 can be used as other input port

P5#: definition of holder form

P47#: delay between holder positive and negative reversion. It’s suitable for bigger holder.

P79#: maximal time of holder positive rotation, if the holder rotates positively after the time defined by P79 #and hasn’t found the target holder, it can be thought that the holder positive rotation is locked or the cutter signal detection is abnormal and the system will prompt 44# alarm.

B157=1: after tool-change is completed, confirm whether current cutter number is the expected one; =0: No longer confirm.

For the tool-change mechanical control P5#=2 or above of other forms, please contact the manufacturer to get related data.

Face plate sequence tool-change key:

Sequence tool-change key on the face plate: change tool in turn according to cutter number defined by P2#, if the system doesn't check the cutter number, this function will be invalid. If tool array (P5#=0) is selected, the system will regulate cutter compensation number and coordinate in turn..

5.4.8 Machine tool alarm processing

One. Limit

The error alarm is caused because each coordinate axis movement exceeds the setting scope (over travel), and the hardware over travel (caused by movement of mechanical upper limit bit switch) and software over travel (exceed the planned maximal scope of coordinate).

After limit occurs, the system will implement the following movement course.

1. Each coordinate axis reduces speed to zero2. Switched to manual feed mode.3. Close spindle and cooling4. Prompt "Error 40" alarm After limit occurs, this axis won't move along this direction, but the reverse movement is permitted to exit the limit state.

The numerical control system can only define positive direction/negative direction limit, namely various axes jointly use a positive direction limit input and jointly use a negative direction limit input; once the positive direction (or negative direction) limit of certain axis occurs, other axes won't move in this direction until the limit is exited.

For the input of hardware over travel, the input number will be defined by parameters: the positive direction limit input number is P72#, and the negative direction limit input number is P73#.

Two. Driving alarm

1～3 processing steps are same as limit

4. If B196～B198 set the internal contracting brake valid, all the internal contracting brake signals of each axis will be cancelled.

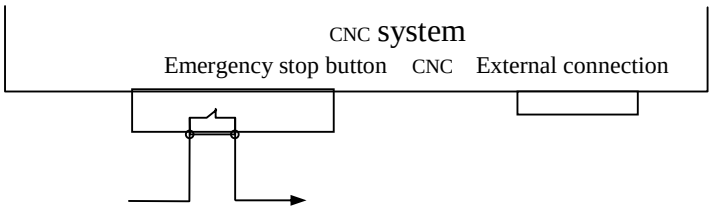
The driving alarm is connected to system defined by XERR, YERR and ZERR of the motor control signal socket.

Three. Emergency stop alarm

The emergency stop function is used to switch off the operating state of machine tool system.

The emergency stop button is provided with two pairs of contacts, a normally opened contact and a normally closed contact; the input signal connected to the system with normally open contact type gives 55# alarm, and the operating state between feeding and spindle will be cut in normal condition and block all operations.

The system won't process the normally closed contact and only introduces to the output terminal, and the user should connect it to circuit of control circuit in strong power cabinet, in case of emergency, switch off the power of movement part compulsorily, ensuring that the machine tool system can stop operation in any case. The voltage and current of this output terminal shouldn't be higher than the indicated upper limit (see back of system)



Emergency stop processing:

1～3 processing steps are same as limit

4. Display "Error 55"5. Lock the keyboard until the emergency stop signal is cancelled6. The pair of normally closed contact at the back of system can be connected to the control circuit in the machine tool electric cabinet which is used to cancel some control circuit in it; the voltage of this contact switch is limited to 24V～36V (AC or DC)**Four. Reset button**

It's the CPU hard reset signal of system below software V5.0.It's soft reset for software 5.0 and the one above, which is used to cancel current state, see description of this version.

Five. General external alarm

The user can connect this alarm signal according to the mechanical structure requirement, and this signal can be determined by B142 whether it's as alarm prompt or processed as severe alarm.

B142:=0: Only "Error 42" is prompted when giving alarm without any operation. =1: processing steps are same as limit 1～3

The access terminal of general external alarm input signal is defined by P74#.

Six. Internal contracting brake control

B196～B198 define whether the internal contracting brake functions of Z-axis, Y-axis and X-axis are open or not, when certain one parameter is set for 1, the corresponding axis will open the internal contracting brake function, and the internal contracting brake control signal (power-on signal) is output by TRF of each axis (XTRF, YTRF and ZTRF); it'll drive a relay and connect power of internal contracting brake, to make the braking of internal contracting brake invalid.

Once the internal contracting brake is defined, it'll output internal contracting brake signal after system sends enabling or monitors the servo ready signal and delays the time defined by Pa4#.

Once the driving of certain on axis gives alarm, the system will switch off all internal contracting brake signals.

5.5 Generation and recovery of work piece coordinate system

The system is provided with two-level coordinate system, which is the machine tool coordinate XP, YP and ZP, and the zero is mechanical origin, the work piece coordinate is X, Y and Z (coordinate displayed with big character); the origin is established in trial cutting and preset cutter and saved in the data system. The coordinate values of two coordinates indicate distance of tool nose point of preset cutter of current cutter to coordinate origin.

The machine tool is established when returning to mechanical zero, while the work piece coordinate is calculated by system during the trial cut preset cutter course, and the work piece coordinate system can be automatically recovered through returning to mechanical zero. For the machine tool without machine tool zero, it can be memorized by the system memorizer. Accordingly, it's strongly suggested to adopt mechanical origin device.

5.5.1 Generation mode of work piece coordinate system

In principle, the coordinate system of numerically-controlled machine tool (machine tool coordinate and work piece coordinate) should recover the work piece coordinate through returning mechanical zero when the machine tool is powered on; however, owing to economic numerically controlled operating habit in actual operation, lots of machine tools won't install the machine tool zero sensor, accordingly, the numerical control system is provided with two modes.

1. Return to machine tool zero after being powered on

After being powered on, the mechanical zero can be only found through selecting "Machine tool zero" mode to manually press direction feeding key, recover the work piece coordinate system, then the modes of automatic processing and MDI will be valid.

2. The machine tool doesn't return to machine tool zero after being powered on

The machine tool can operate after starting up by depending on the coordinate information which is saved in the power down protective memory, the work piece coordinate and the machine tool coordinate are the same as the condition before previous shutdown; because the motor tool carriage is in free state after shutdown or the motor over charging is caused owing to sudden power down when the machine tool is switched off last time, so that the coordinate recovered by system doesn't conform to the actual position, and failure occurs.。

5.5.2 Generation of machine tool coordinate and work piece coordinate

1. Zero-returning mode in power-on (B033=0)When the system is powered on for the first time, various coordinates and cutter information are in the undefined state, different data must be generated through the following sequence:

(1) Cutter clamping: at least one holder (or tool array holder) is installed on the holder as current cutter

(2) Start machine tool and switch power on

(3) Memory zero clearing (adopt the memory clearing function in primary function)

(4) Each shaft returns machine tool zero

(5) Perform trial cut and preset cutter, to generate workpiece coordinate system of current cutter

(6) Other cutter respectively presets cutter

(9) Switch off the power and then switch on the power, return zero of machine tool, and the system is recovered to the workpiece coordinate of current cutter at the zero of machine tool, and the workpiece coordinate will be valid

2. No zero-returning mode after power is on (B033=1)The operating step of zero-returning mode must be adopted except the fourth step can be omitted

5.5.3 Parameter options related with coordinate system:

For lathe control:

B033=0: mechanical zero must be returned after power is on

=1: mechanical zero must be returned after power is on

B034=0: recover work piece coordinate according to cutter number

=1: work piece coordinate is cleared to zero

For the lathe mode and electric holder (P5#=1) mode, if the system doesn't check the effective cutter number signal at the mechanical zero position, the work piece coordinate will be cleared to zero. For tool array, the cutter number is set for zero number, and the cutter compensation number is set for 1 after power is on, and recovers 1# cutter work piece coordinate. For the holders with other form, the user must appoint the cutter number generating mode after power is on.

Under the no zero-returning mode after power is on, the cutter compensation number of tool array will be generated by system.

For the milling machine system, three axes coordinate includes two options after returning zero:

B033=0: mechanical zero must be returned after power is on

=1: mechanical zero must be returned after power is on

B034=0: P21# parameter recovers X-axis work piece coordinate.

P22# parameter recovers Y-axis work piece coordinate.

P75# parameter recovers Z-axis work piece coordinate.

B034=1: the work piece coordinate of each axis at X Y Z mechanical origin is cleared to zero

5.5.4 Coordinate conversion G54-G57

The system is in G54 state after power is on, the work piece coordinate can be recovered from the effective information stored in the system when returning to machine tool zero; however, other coordinate conversion G55-G57 set by system is slightly different from traditional coordinate conversion, which is convenient for user to implement the coordinate conversion operation.

G54: recover work piece coordinate when power is on

G55: absolute value work piece coordinate conversion: regard the position of coordinate value after G55 as the zero of new work piece coordinate value

G56: relative value coordinate conversion: regard the coordinate value after G56 as corresponding current offset, and the position after offset is the new work piece zero

G57: current point bias: regard the current position of cutter as new coordinate zero to set coordinate system

After the automatic program is executed, automatically execute G54 to recover initial work piece coordinate system, preventing confusion

Under MDI mode, G55-G57 is only valid in program

5.5.5 Processing start position setting

It's convenient for operator to move cutter to certain fixed position as start point of processing; the system sets two modes to move the cutter to this position:

G75: return processing start point from the machine tool coordinate value

G76: return processing start point from the machine tool coordinate value

For G75: the system parameter P8#, P24# and P9# are respectively the coordinate values of processing start points of X, Y and Z axis under the machine tool coordinate

For G76: the system parameter P18#, Pb4# and P19# are respectively the coordinate values of X, Y and Z axis under work piece coordinate

The operating results of G75XYZ or G76XYZ are respectively operated to the following coordinate

	X	Y	Z
G75	P8#	P24#	P9#
G76	P18#	P22#	P19#

5.5.6 Cutter compensation modification and cutter bias

The practical cutter will appear two conditions during the processing which will influence the coordinate value and processing size, thus the following measures must the taken to regulate the coordinate

1. Micro-abrasion of cutter Adopt cutter compensation modification: the micro-modification value is added with cutter compensation value or subtracted from the cutter compensation; it depends on the digit parameter B151

B151=0: modification quantity is added with cutter compensation value =1: modification quantity is subtracted from the cutter compensation value

2. Tool nose crack Dismount the cutter from the tool holder for sharpening or replacement, and the tool nose coordinate is changed much, and it's only to perform trial cut and preset cutter to establish the cutter compensation value and corresponding work piece coordinate; however, this function is implemented under the following premise

(1) Return zero mode after power is on

If this cutter is used to return to mechanical origin when power is on, it must have been preset.

(2) No zero-returning mode in power-on

From previous presetting cutter to re-presetting cutter, one of the following functions can't be implemented:

a) System memory zero clearing

b) Coordinate bias is executed

c) Work piece coordinate system is set again after the motor locked rotor and block

When the condition above can't be confirmed, it's suggested that the user should preset cutter again when replacing other cutters

3. Entire offset of holder The servo-motor causes entire offset of coordinate owing to locked rotor and block, which causes damage of work piece coordinate system:

(1) Return zero mode after power is on

It's only necessary for user to return each axis to mechanical zero, and the work piece coordinate system will be automatically recovered

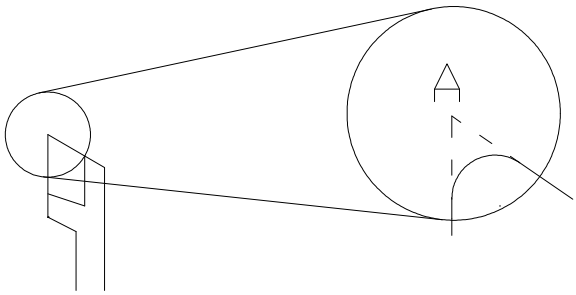
(2) No zero-returning mode in power-on

If the work piece size is changed in X and Z direction, the coordinate bias function can be used to input the change value in X and Z direction; input the change value of work piece size, input positive value if the work piece size is increased and input negative value if the size is reduced.

5.6 Tool nose radius compensation

5.6.1 Overview

When we use trial cut and preset cutter, the work piece will be cut in X and Z direction respectively; establish the work piece coordinate after a series of operation, in general, after the trial cut and preset cutter are performed in X and Z direction, the coordinate will indicate the A point coordinate of tool nose of cutter (as Figure 5-4); however, the practical tool nose will have a segment of R arc, causing that A point is an assumed point in fact.

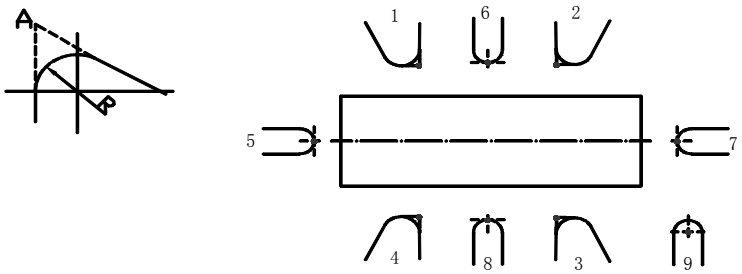


F5-1

When the combined movement cutting in X and Z direction is performed, the outline error will be caused

5.6.2 Phase definition of tool nose

Because the preset cutter mode of tool nose and the practical shape of cutter are different from the relative position of work piece, the cutter will be different from the relative position of work piece as Figure 5-5.



F5-2

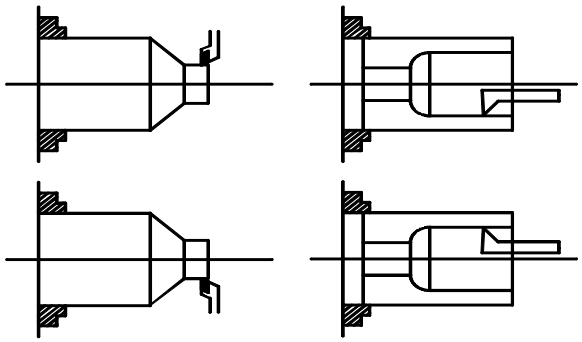
There are 8 kinds which are provided with 1 to 8 phase, and there is a holding condition, namely the preset cutter point of tool nose is in the acc center of tool nose (for instance, this position will be probably preset by adopting optical tool setting gauge), which is defined as 9# phase

The information above must be memorized in the cutter table

5.6.3 Cutter parameter table

Each cutter has 4 parameters, DX and DZ is the cutter compensation value, R is the tool nose radius and PH is the phase of this cutter.

5.6.4 Track direction definition of tool nose compensation



F5-3

5.6.5 Process of establishing and canceling cutter compensation

For the lathe, after one cutter is preset, the work piece coordinate is only the coordinate corresponding to coordinate origin of A point on the tool nose

During the course of cutter compensation, the coordinate movement is still the A point coordinate, unless to modify the coordinate system through modifying the instruction of coordinate system to move the work piece coordinate to other place.

The cutter compensation course is composed of three stages

1. Establish cutter compensation; 2. Cutter compensation course; 3. Exit cutter compensation As a matter of convenience, the description later can regard the cutter point shape as the part circle or full circle of radius R

1. Establish cutter compensation It can only regard G01 as the statement of establishing cutter compensation, to make system enter the cutter compensation course from no-tool cutter compensation state, and the coordinate system in the system implement proper regulation and the representative statement is as follows:

G42 T03 F100

G01 X30 Z100

or G42 G01 X30 Z100 T01 F100

Track description:

For the condition that the angle is less than 180

1. Next track is straight line 2. Next track is arc

（a） F5-4 （b）

For the condition that the angle is greater than 180

（a） F5-5 （b）

The movement course of tool nose during the cutter compensation course

1. The angle is less than 180

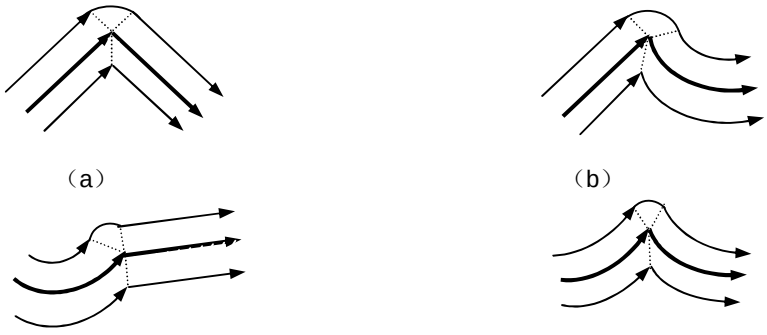
(a) (b) (c) (d)

F 5-6

2. The angle is larger than 180

（a） F5-7 (b)

3. The movement course of tool nose during the cutter compensation course



F 5-8

The system will give alarm when some special conditions such as cross of track can't be handled.

5.7 System software upgrade

All the WASHING CNC systems can be upgraded through USB disk and serial port. System upgrade can be executed without opening the machine box on the working floor. The upgrade will not effect the original parameter file, machine tool coordinate system , cutter compensation and machine tool status information, that means the operations to re-establish parameter file, cutter compensation, etc, are not needed when finished upgrade, including boot up interface.

5.7.1 When system software up grade is needed

1. Customized upgrade software with new function.
2. The system gained new function.
3. System software renewal

5.7.2 How to get upgrade software

The system upgrade software name of WASHING company is WXSYS, you may get the upgrade software through the following methods:

1. Directly provided by technical service of WASHING company.
2. Provided in email from Nanjing WASHING CNC Technology Co., Ltd

Note: Upgrade software files known as:

31XTA：**S31XTA** 32XTA：**S32XTA** 300T：**S300T**

99TA：**S99TA** 99TB：**S99TB** 99TY：**S99TY** 99UZ：**S99UZ**

5.7.3 How to enter system upgrade interface or download user picture interface

Press hardware reset button to reset system(hardware reset button is beside the USB disk port, you can find it when open USB disk port cover)，and press button ALT **before pressing system hardware reset button, or press** ALT **button when power-on system, until it popup input box "please input password", then release** ALT **button. Input password in the input box (initialized password: GGG), and press ENTER, the system will popup system upgrade interface.**:

USB: press this button to select upgrade through USB disk

serial port: press this button to select upgrade through serial port.

system software: press this button to select upgrade system software.

user interface: press this button to select renew user interface

CHN font: this button to select renew CHN font

5.7.4 How to upgrade system through USB disk

Operation steps below:

1. Get the system upgrade software, see method in Section 5.7.1.2.

2. Save the upgrade software named as WXSYS to USB disk root directory

3. Enter system upgrade interface

4. In system upgrade interface, press F1(USB)，and press F3（system software），the two buttons showing as pressed status, see illustration 1.3

5. Press F8（confirm），the system starts operation to upgrade, and popup system upgrade progress interface, see illustration 1.4，and showing current system upgrade steps until the interface prompts information as ”system upgrade finished, please restart system” that means the system upgrade is accomplished.

6. restart system, check whether the system can proper start or not, if it can proper start, it means the system upgrade successfully. You may operate the machine tool for some simple test, such as: cutter change, etc, if test passed, the system may carry out trial processing operation, if trial processing passed, the system may execute normal processing operation.

5.7.5 User boot up interface renew

The system provides a certain space to store user boot up interface，you may download the picture to the CNC system through USB disk or serial port, and the system will display the renewed picture when boot up in the future. The maximum system supporting picture size is 480×320 pixel, the system supporting 256 color.

5.7.5.1 How to edit their own user interface boot

You may use the picture tools in the Windows operation system to edit pictures, and save the edited picture to format as 256 color，480×320 pixel，Upon completion of editing, Save the file, Custom file name

When using U-boot disk for the screen upgrade:

31XTA：**U31XTA** 32XTA：**U32XTA** 99UZ：**U99UZ**

99TA：**U99TA** 99TB：**U99TB** 99TY：**U99TY**

Operation steps below:

1. Get the system upgrade software, see method in Section 1.2.

2. Save the upgrade software named as WXSYS to USB disk root directory

3. Enter system upgrade interface

4. In system upgrade interface, press F1(USB)，and press F4（system code），the two buttons showing as pressed status, see illustration 1.3

5. Press F8（confirm），the system starts operation to upgrade, and popup system upgrade progress interface, see illustration 1.4，and showing current system upgrade steps until the interface prompts information as ”system upgrade finished, please restart system” that means the system upgrade is accomplished.

6. restart system, check whether the system can proper start or not, if it can proper start, it means the system upgrade successfully. You may operate the machine tool for some simple test, such as: cutter change, etc, if test passed, the system may carry out trial processing operation, if trial processing passed, the system may execute normal processing operation.

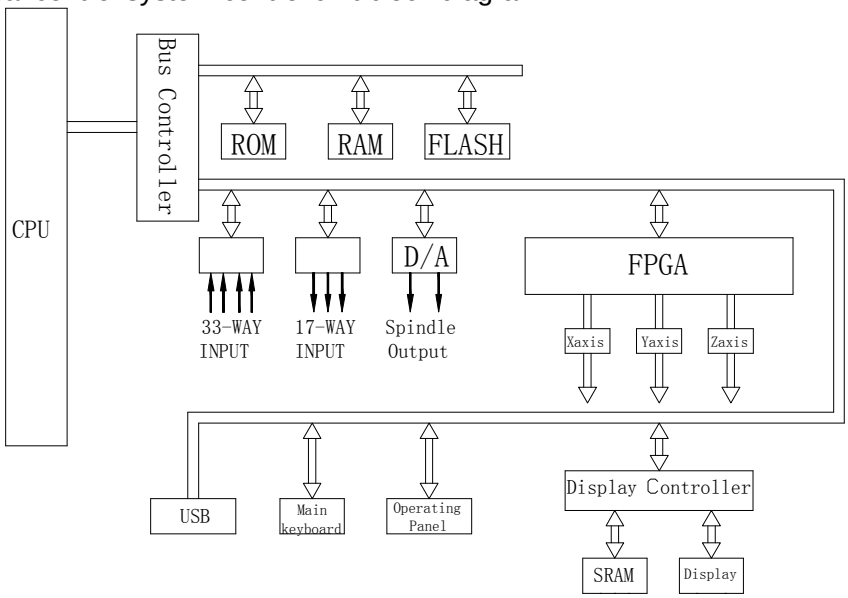
5.7.5.2 How to renew user interface through serial port

Save the generated file of USERPIC into USB disk root directory, the operation process to renew user interface is the same as the operation process to upgrade system software, change the operation of selection “system software” to selection “user interface”. Or in the system upgrade interface, press F4（system code）to upgrade system file, press F5（user interface） to upgrade user interface, and the other operations are the same.

Chapter six System connect

6.1 System composition

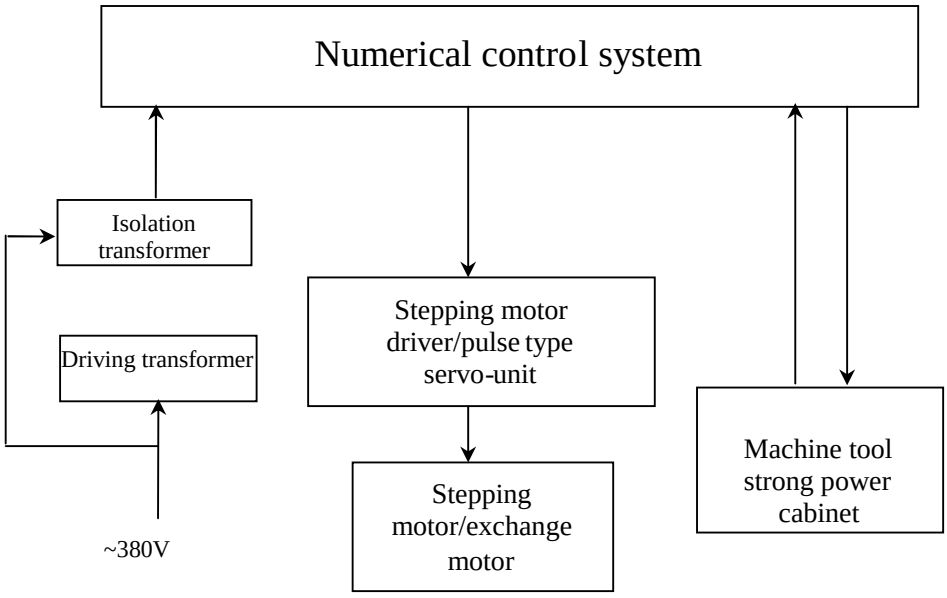
6.1.1 Numerical control system control unit block diagram



F6-1

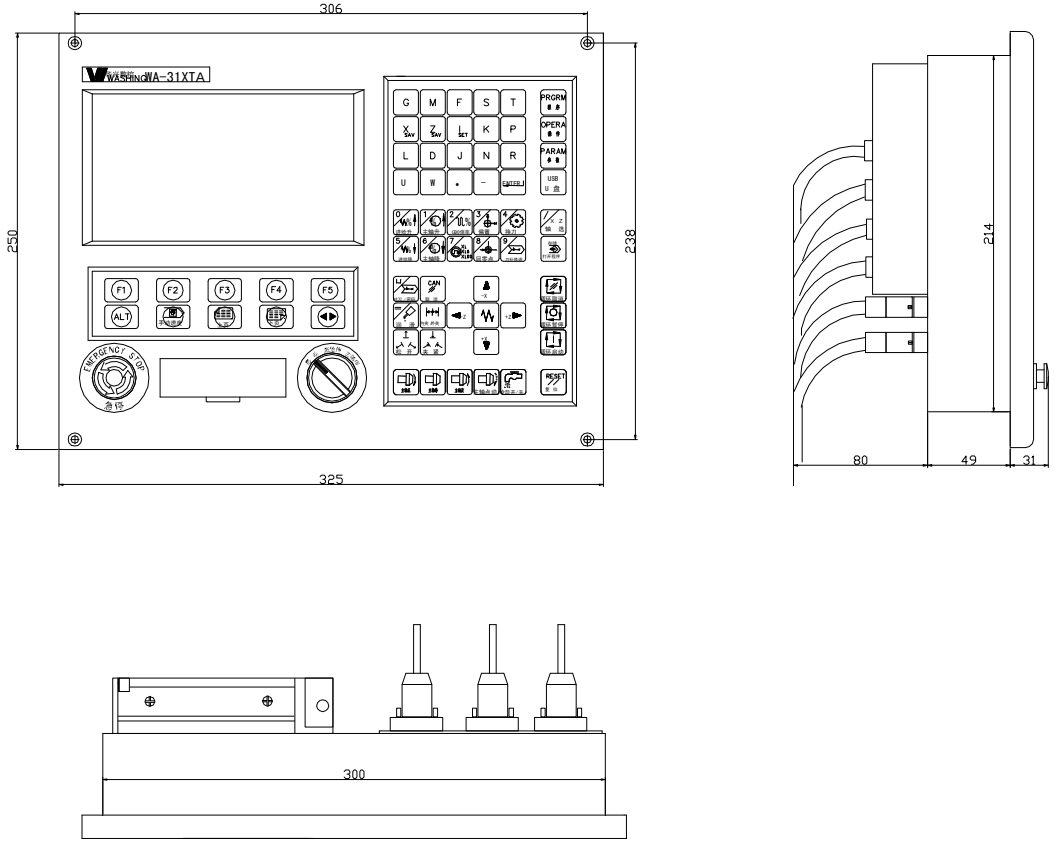
6.1.2 The machine tool numerical control system composed of numerical control system should include the following contents:

(1) CNC control unit and accessories
(2) Stepping motor drive driving power/pulse servo unit
(3) Stepping motor/Servo motor
(4) Machine tool power closet

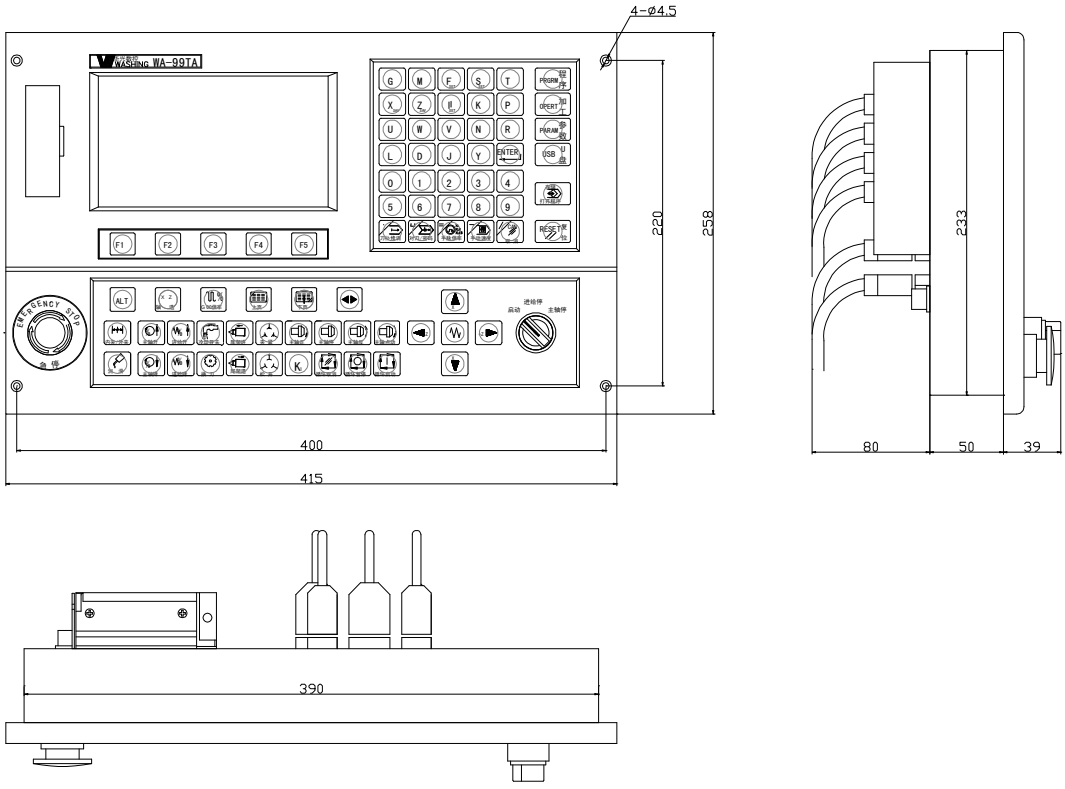


F6-2

6.1.3 Dimensions



（a）31XTA/32XTA

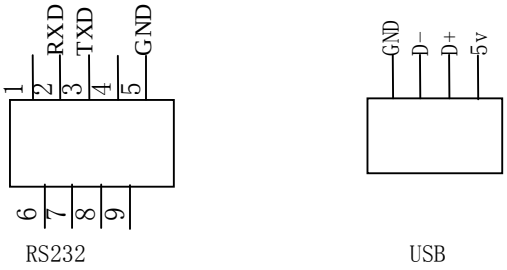


(b)99TA/99TB/99TY

F6-3 **Dimensions**

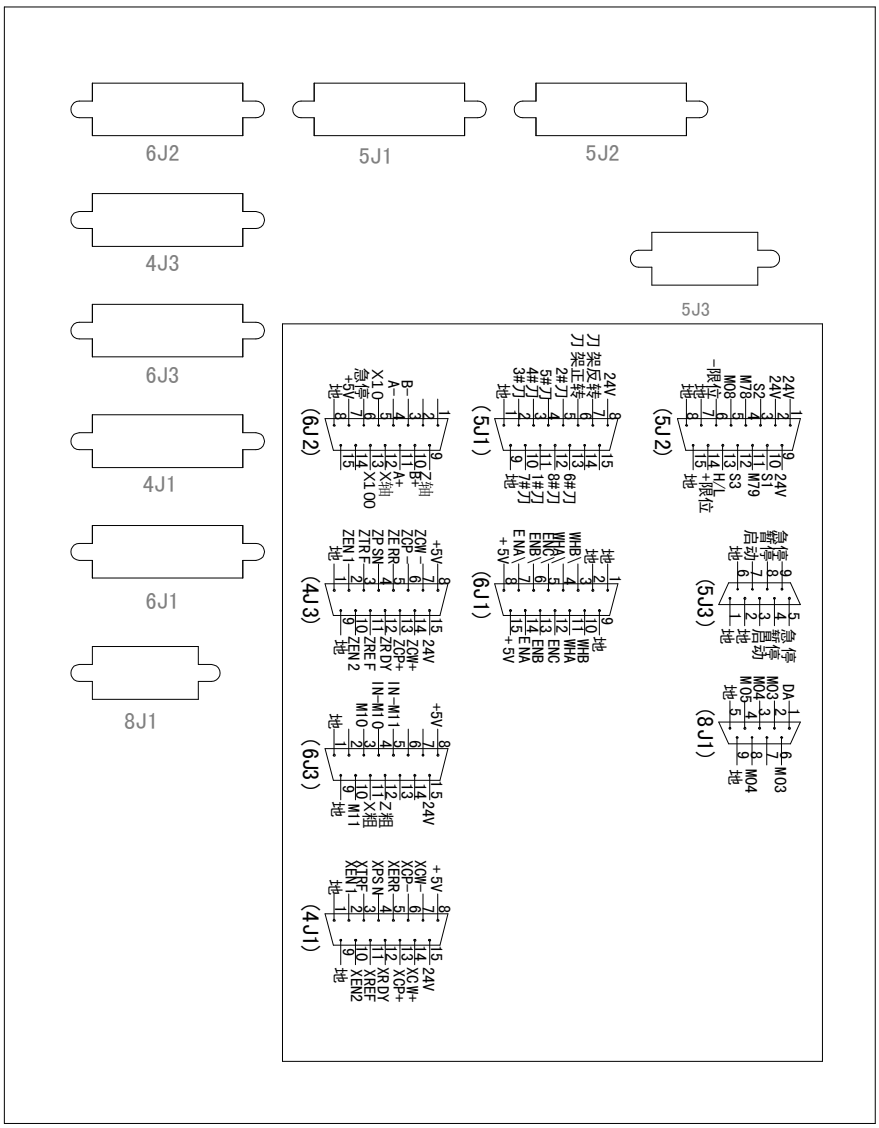
6.1.4 Port definition list

6.1.4.1 Front cover panel(in plastic box cover)definition：



F6-4

6.1.4.2 Back cover panel port definition



F6-5

6.1.5 Output signal comparison list

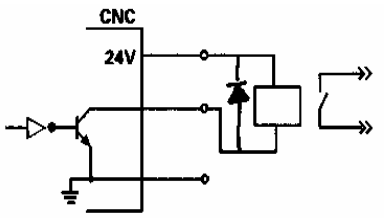
T6-1 31XTA/32XTA/99TA/99TB/99TY/99UZ/300T Output signal comparison list

Pin and No.	Pulse output	Retain output	Cancel output	Default function	Remarks
4J1-P2（P10）	M85	M20 K15	M21 K15		X axle ENABLE，contact point output
4J1-P3	M84	M20 K14	M21 K14		X-axis switch off and on output
6J3-P10		M20 K13	M21 K13	M11	Spindle loosening output
6J3-P3		M20 K12	M21 K12	M10	Spindle clamping output
4J3-P2（P10）	M81	M20 K11	M21 K11		Z axle ENABLE，contact point output
4J3-P3	M80	M20 K10	M21 K10		Z-axis switch off and on output
5J1-P6	M75	M20 K4	M21 K4	Cutter rest clockwise rotary	Power output
5J1-P7	M74	M20 K5	M21 K5	Cutter rest countercloc kwise rotary	Power output
5J2-P3		M20 K16	M21 K16	M15/S2	S1，S2，S3 interlock
5J2-P4	M78	M20 K8	M21 K8		Power output
5J2-P5	M76	M20 K6	M21 K6	M08	M09 cancel M08，Power output
5J2-P10		M20 K17	M21 K17	M14/S1	S1、S2、S3 interlock
5J2-P11	M79	M20 K9	M21 K9		
5J2-P12	M77	M20 K7	M21 K7	M16/M12/S 3	S1、S2、S3 interlock, Power output
8J1-P2（P6）	M71	M20 K1	M21 K1	M03	M03、M04 interlock，contact point output
8J1-P3（P8）	M72	M20 K2	M21 K2	M04	M03、M04 interlock，contact point output
8J1-P4	M73	M20 K3	M21 K3	M05	M05 cancel M03 、M04 interlock, Power output

Explanation：

Contact point output：a relay in CNC output contact point signal and the contact point can stand voltage ≤36V，currency ≤500mA，it is forbidden to connect control power that higher than this power into internal contact point.

Power output: it is power audio output within CNC to drive external inter relays.



6.1.6 Input signal comparison list

T6-2 31XTA/32XTA/99TA/99TB/99TY/99UZ/300T Input signal comparison list

Pin and No.	Terms input	Default function	remark（default function）
4J1-P4	M01 L14/K14	XPSN	X axle exact stop（at arrival）
4J1-P5	M01 L16/K16	XERR	X axle alert
4J1-P11	M01 L26/K26	XREF	X-axis reference point servo Z impulse input
4J1-P12	M01 L15/K15	XRDY	X axis servo gets ready to input
6J3-P4	M01 L10/K10	INM10	Main axle holding
6J3-P5	M01 L12/K12	INM11	Spindle loosening input
6J3-P11	M01 L27/K26	XRGH	point is defined by 66# P parameter

Pin and No.	Terms input	Default function	remark（default function）
6J3-P12	M01 L11/K11	ZRGH	point is defined by 68# P parameter
4J3-P4	M01 L2/K2	ZPSN	Z axle exact stop（at arrival）
4J3-P5	M01 L1/K1	ZERR	Z axle alert
4J3-P11	M01 L28/K28	ZREF	Z-axis reference point servo Z impulse input
4J3-P12	M01 L9/K9	ZRDY	Z axis servo gets ready to input
5J1-P2	M01 L17/K17	3# cutter	3# cutting tool signal input
5J1-P3	M01 L21/K21	4# cutter	4# cutting tool signal input
5J1-P4	M01 L22/K22	5# cutter	5# cutting tool signal input
5J1-P5	M01 L24/K24	2# cutter	2# cutting tool signal input
5J1-P10	M01 L23/K23	7# cutter	7# cutting tool signal input
5J1-P11	M01 L20/K20	1# cutter	1# cutting tool signal input
5J1-P12	M01 L19/K19	8# cutter	8# cutting tool signal input
5J1-P13	M01 L18/K18	6# cutter	6# cutting tool signal input
5J2-P6	M01 L4/K4	LIM-	Each shaft negative limit input, defined by 73# P parameter
5J2-P13	M01 L5/K5	H/L	Main axle high/low speed gear signal input
5J2-P14	M01 L3/K3	LIM+	Defined by 72# P parameter
5J3-P3, 5J3-P6	M01 L6/K6		External startup
5J3-P4, 5J3-P7	M01 L7/K7		External pause
5J3-P5, 5J3-P8	M01 L8/K8		External emergency stop
6J2-P5		×10	external hand wheel ratio “X10”
6J2-P13		×100	external hand wheel ratio “X100”
6J2-P6			Hand-wheel emergency stop input
6J2-P12		X axis	Hand-wheel X-axis selection input
6J2-P9		Z axis	Hand-wheel Z-axis selection input

Note：MO1 L__;M01 K__

L is active low level, K is active high level

The number after L or K is the input number

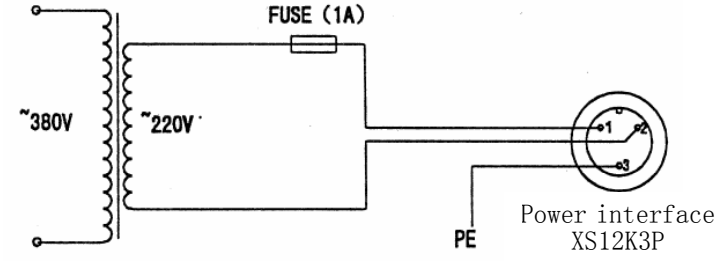
6.2 Heavy current power supply

6.2.1 Installation requirement

WASHING CNC system should work in good mechanical and electric environment, it should be properly installed in mechanical and electric consideration and the input/output ports should be properly connected. To CNC system, you should make a box to install the system, there are six Φ4.5 through holes in the system panel to fix the system to the box with M4 screws. The box size should be big enough to take the length of plug behind the system and wires into consideration. The box should have good heat sink character.

6.2.2 Heavy current power supply

Washing CNC system requires the voltage of power supply variation stands within more/less 10% of standard nominal voltage. So we suggest to install a 150VA isolated transformer. See Illus 7-7.



Illus. 7-7

Note: the output voltage of transformer mentioned in this manual are all no-load voltage, the capacity can not be lower than the specified value.

6.2.3 Earth

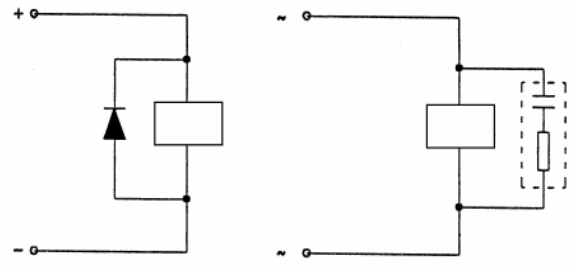
Earth is very important in electrical installation. Proper earth can make the CNC system operate more steady and reliably and avoid electricity leakage. Washing CNC system has external earth point and the point must be reliably connected to earth during application. Live up to:

1. Make sure the whole machine tool electrical system must be connected to a main earth point and properly earthed.
2. The signal earth of electronic device that has communication with CNC system must be connected to earth point and the earth point must be properly connected to the main machine tool earth point, the connection wire area no less than 2.5mm^2.
3. The signal wire must have shield layer, for it can have better anti-jamming effect if to use the shield layer as power earth transfer.
4. It is forbidden to use the A/C earth wire (middle line in the three phase currency) as earth wire.

6.2.4 The issues need attention during H/C installation

Washing CNC system must be connected with machine tool heavy current circuit to control the whole machine tool actions. To make use the system working properly, all induction load of heavy current part of machine tool should be installed with interrupter devices. Suggestions below(illus. 7-8):

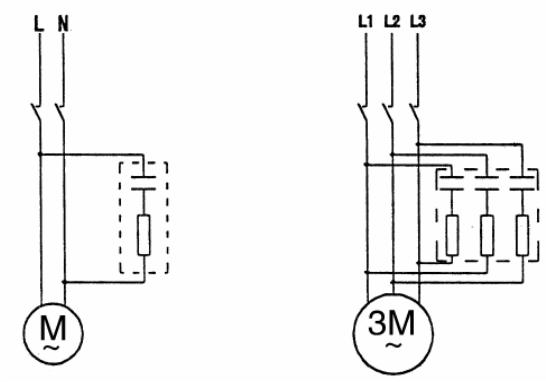
1. To A/C relay loop, install single phase interrupter that is parallel connected to the ends of connector loop.
2. To D/C relay loop, parallel connect diode to transfer currency.



(a) D/C relay loop (b) A/C relay loop

Illus.7-8 interrupter devices

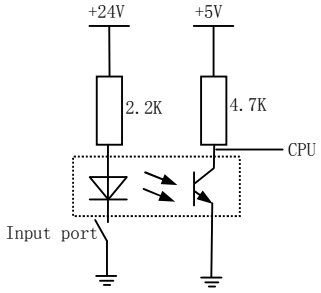
A/C motor：Install single phase/three phase interrupter device according to the motor is single phased or three phased, home-made absorbing circuit with separate resistance, capacitance is not permitted. RC must be installed to the load terminal of switch or contactor, See typical connection below:



Illus. 7-9
Internal connection of CNC system

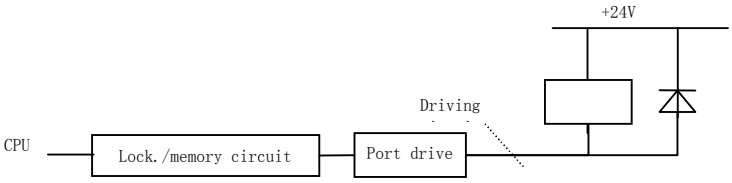
6.2.5 Input/output illustration

1. Input port circuit illustration



Illus7-10 input port circuit

2. Output port circuit illustration

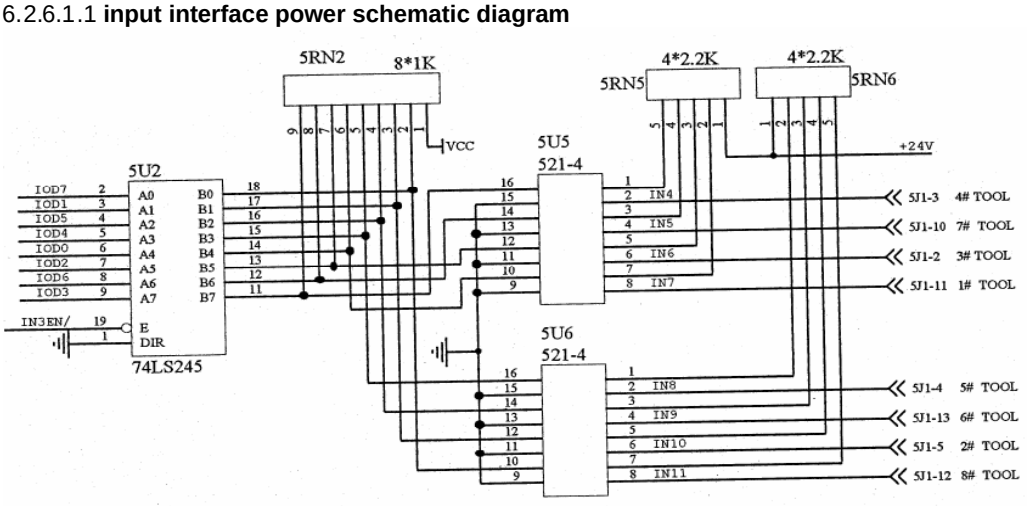


Illus.7-11 Output circuit

6.2.6 Numerical control System Internal Connection

6.2.6.1 input, output schematic diagram

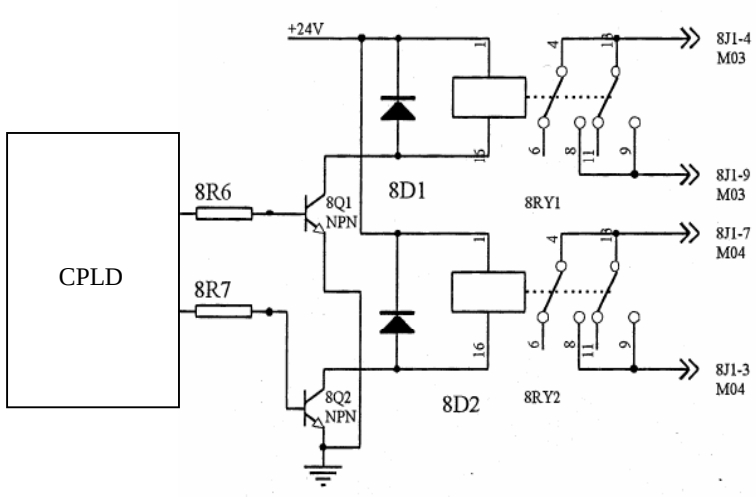
6.2.6.1.1 **input interface power schematic diagram**



F6-12 **Electro-holder interface circuit**

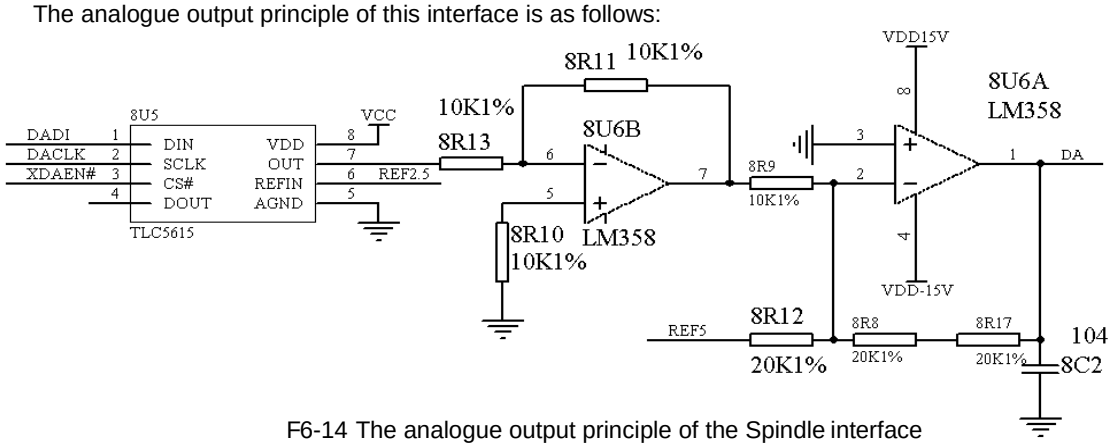
6.2.6.2 Spindle interface

The output principles of this interface M03 and M04 are as follows:



F6-13

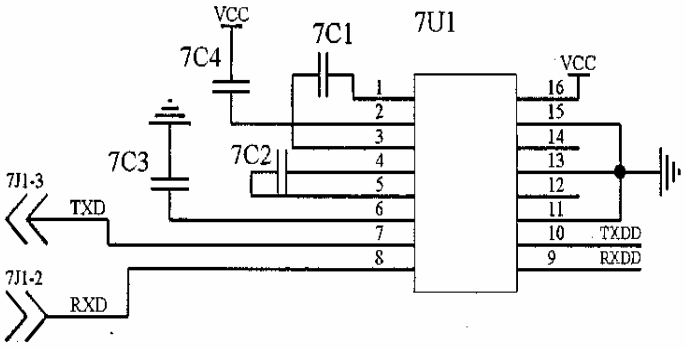
The analogue output principle of this interface is as follows:



F6-14 The analogue output principle of the Spindle interface

6.2.6.3 RS232 interface

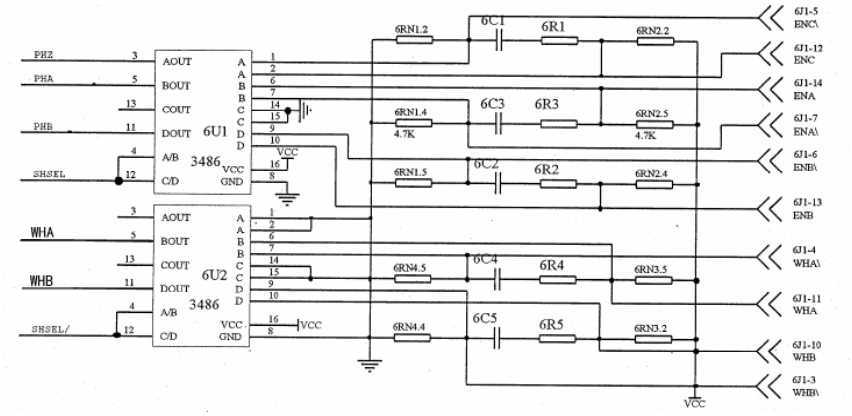
This interface is the simplified asynchronous RS232 port, and the internal principle is as follows:



F6-15

6.2.6.4 Hand-operated pulse generator, encoder interface

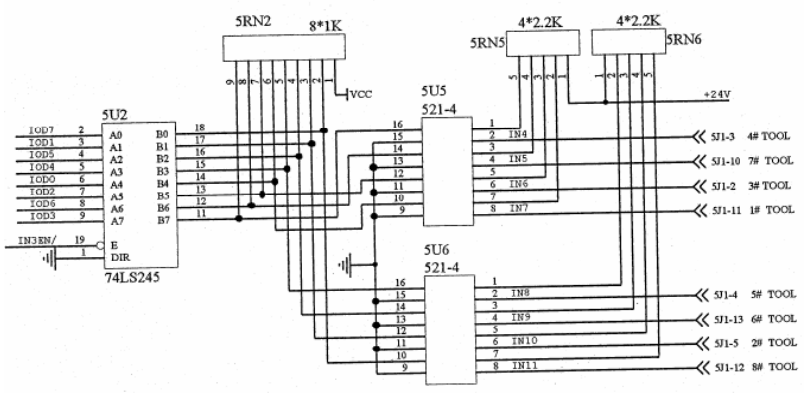
As Figure 6-16, the hand-operated pulse generator and the encoder can't be effective simultaneously in the numerical control system.



F6-16 Hand-operated pulse generator and encoder interface schematic diagram

6.2.6.5 External start, stop and emergency stop interface

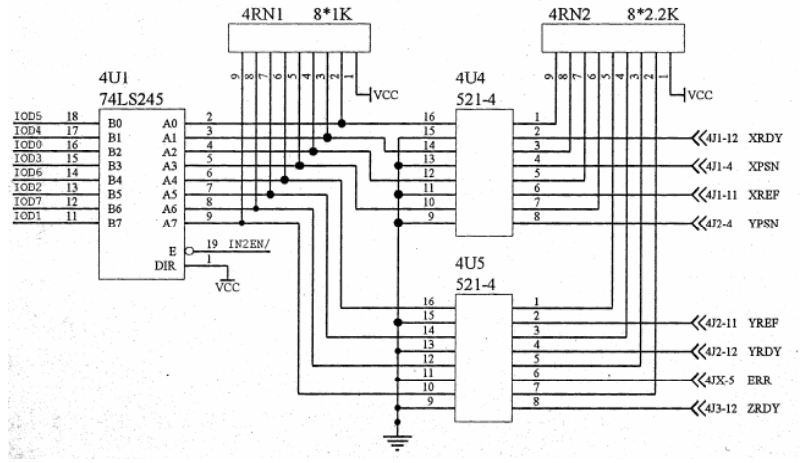
This port is the input port, and the internal principle is as Figure 6-17:



F6-17

6.2.6.6 X, Y and Z axis servo ready, external angle-specified stop, reference point and alarm

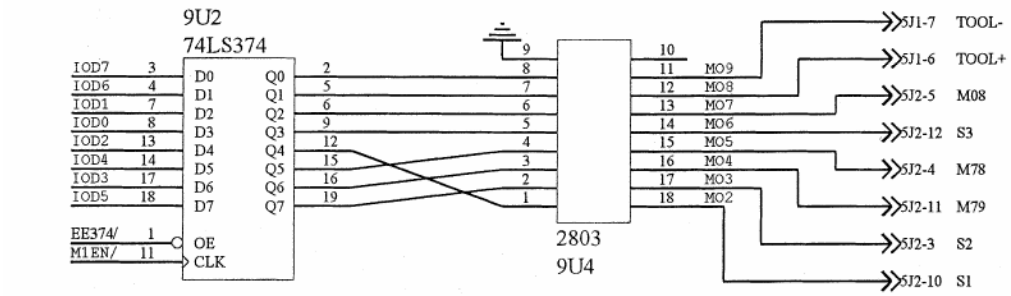
This port is the input port, and the internal schematic diagram is as Figure 6-18:



F6-18

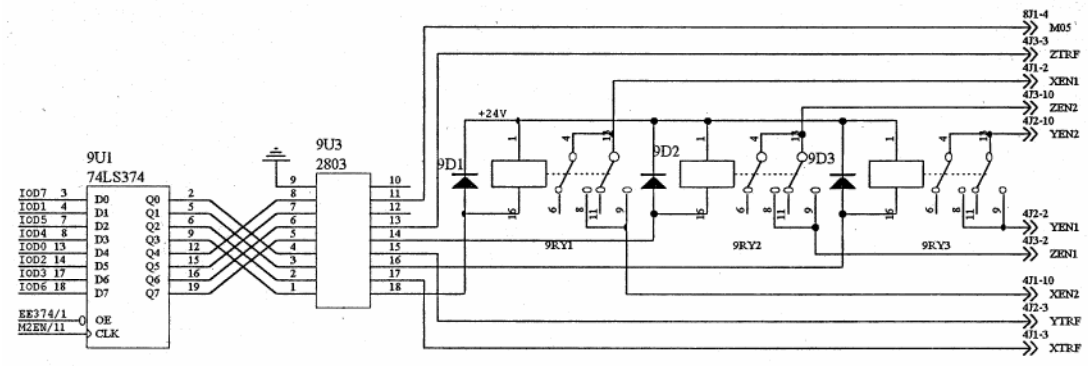
6.2.6.7 Other S, T, M output port

This interface is primarily used to output other signals except main M function such as S1, S2, S3, M79, positive rotation of holder and negative rotation of holder and so on.



F6-19

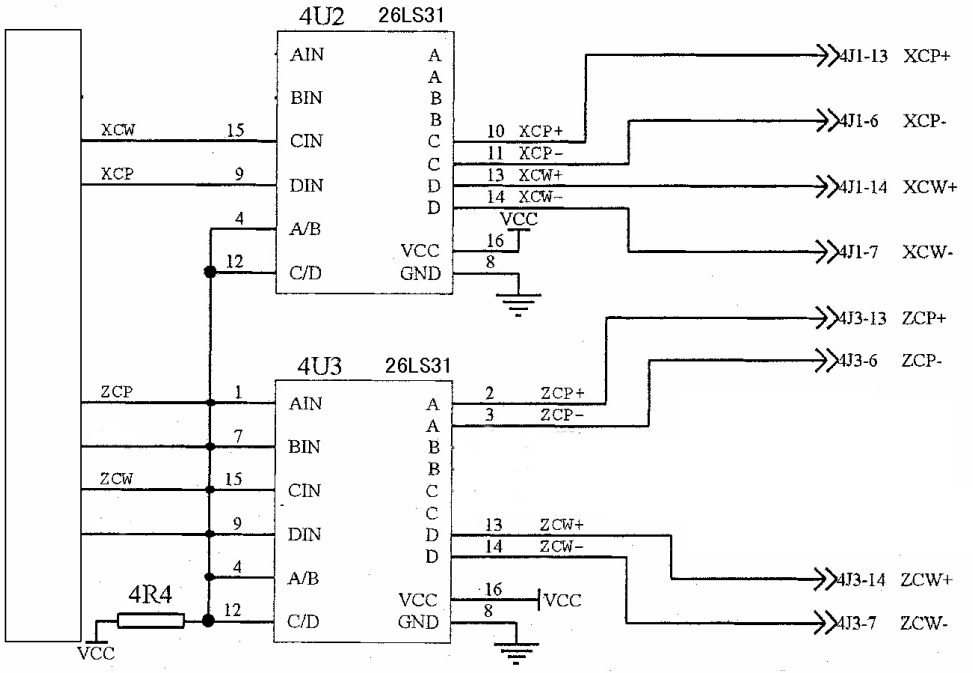
6.2.6.8 Servo enabled, servo-motor zero finding output port



F6-20

6.2.6.9 Motor signal interface

This port is primarily used to output X, Y and Z axis motor driving signal, and each axis has CP and CW differential output signal.

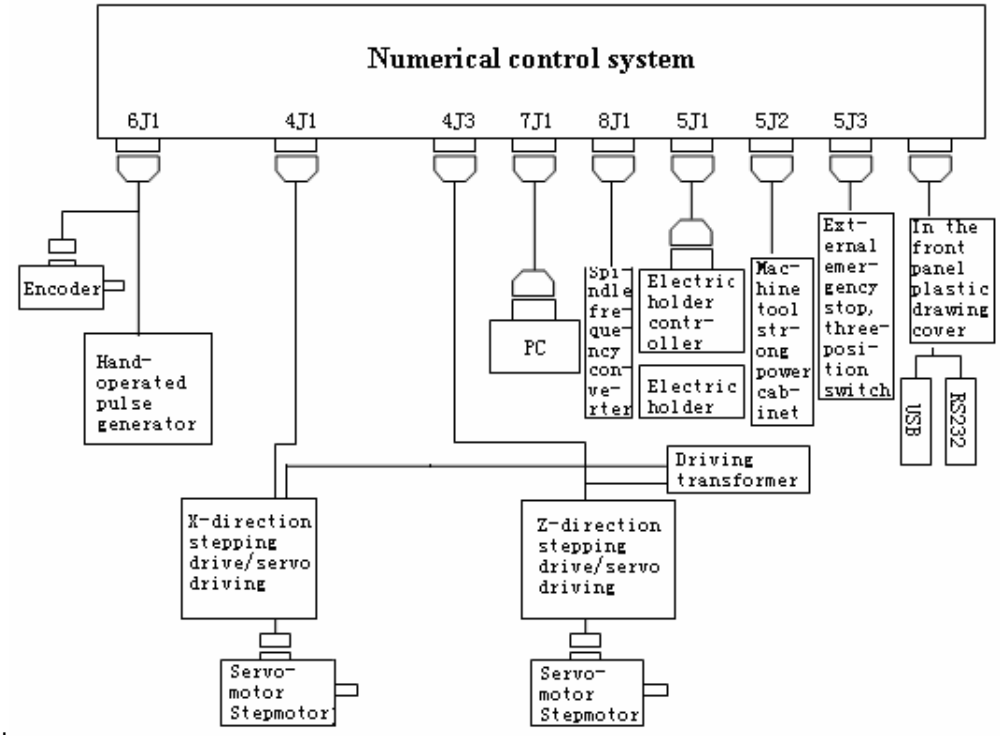


F6-21

6.4 Numerical control system signal interface definition

6.4.1 Numerical control system external connection

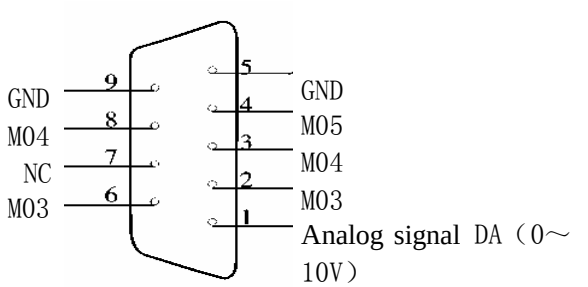
The components related with this CNC control unit are connected as Figure 6-22.



6.4.2 Spindle interface 8J1

The model of this interface is 'DB9 hole', and the plug connected with it should be 'DB9 pin'. The definition is as follows:

P1：Analog signal DA（0～10V）
P2：M03
P3：M04
P4：M05
P5：Analog grounding
P6：M03
P7：NC（Null pin）
P8：M04
P9：Analog grounding



Description: M03 and M04 are the output of relay contact, and the corresponding pin is M03: P2/P6, M04: P3/P8, the voltage of this contact ≤ 36V, current ≤ 500mA, M05 is power output.

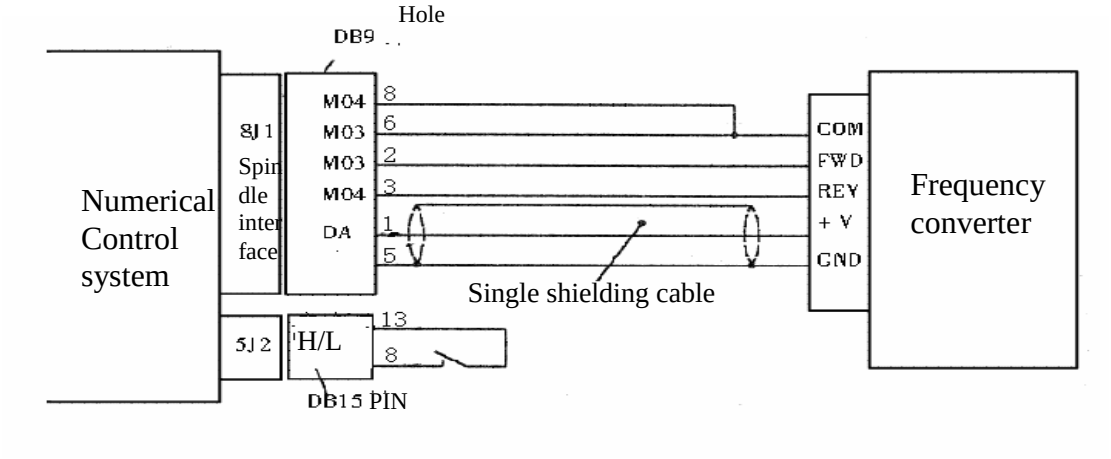
The analog signal (DA) outputs 0～10V analog voltage and is connected with frequency converter. The analog grounding and the signal grounding are mutually connected in the system. This wire must independently use a core shielding wire and the ground wire is a shielding layer. The default value of system is 0～＋10V, which is matched with contact signal to control positive rotation, negative rotation and speed change of the frequency converter. The peripheral equipment (frequency converter) is required absorbing current ＜5mA.

If the spindle is provided with first-level mechanical speed transmission, the exterior should increase one contact as input of high/low speed (13 pin of 5J2), thus the system will distinguish which gear is for the spindle, so as to output proper analog voltage. If the 3# parameter of system is 2000 and 4# parameter is 1000, there will be the corresponding relationship as follows: (it's supposed to be 0～10V gear)

5J2OFP12and GND state / Set spindle speed	Switch off	Switch on
S=2000rpm	DA output 10.00V	
S=1000rpm	DA output 5.00V	DA output 10.00V
S=500rpm	DA output 2.50V	DA output 5.00V

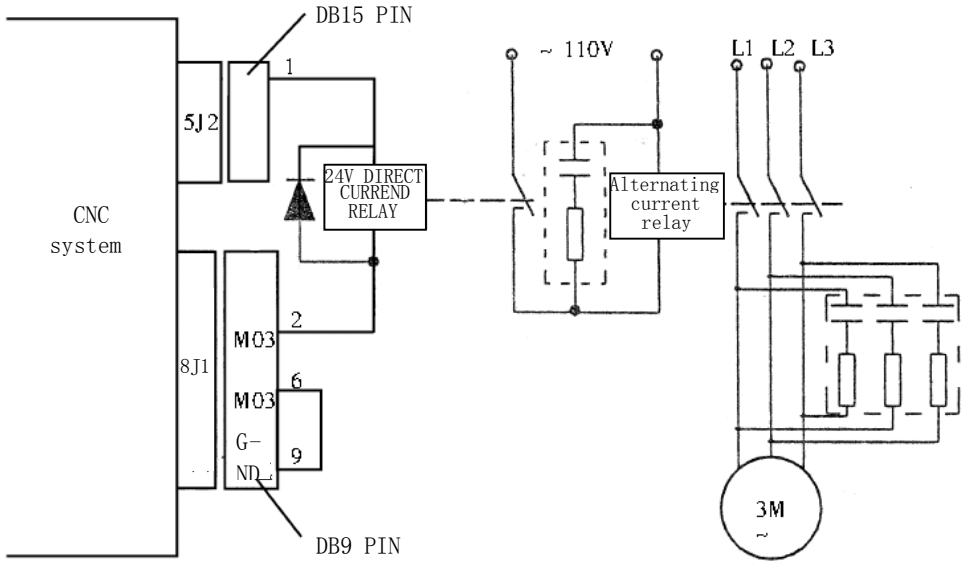
Accordingly, when the spindle is in high gear, P13 of 5J2 should be disconnected with ground; when the spindle is in low gear, P13 should be connected with the ground.

The connection diagram of this interface and the frequency converter is as Figure 6-23。



F6-23

If to directly control the three phase motor clockwise/counterclockwise and stop movement, see reference in the basic circuit diagram in Illus. 7-24.



F6-24

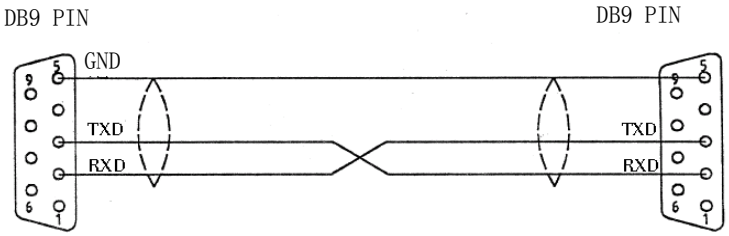
6.4.3 Serial communication port 7J1

The serial communication port 7J1 is the socket of ‘DB9 pin’，the corresponding pin is‘DB9 socket’，to exchange programmer between PC or system, the port definition below:（ not marked pin is empty）：

P1：NC P6：+5V
P2：RXD P7：NC
P3：TXD P8：NC
P4：NC P9：NC

The communication cable should be dual cored isolation wire, and use the shield layer as earth connection wire.

Length≤10M，facture according to F6-25：



F6-25

6.4.4 Tool holder port 5J1

The tool holder port of 5J1 is 'DB15 pin' socket，the corresponding pin is'DB15 socket'，definition to port are below (not marked pin is empty):

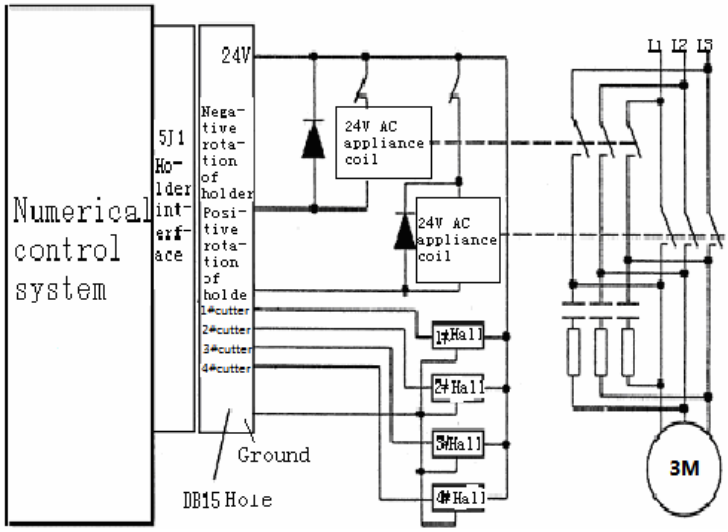
P1：GND
P2：3＃Cutter
P3：4＃Cutter
P4：5＃Cutter
P5：2＃Cutter
P6：cutter holder clockwise rotary
P7：cutter holder counterclockwise rotary
P8：＋24V
P9：GND
P10：7＃Cutter
P11：1＃Cutter
P12：8＃Cutter
P13：6＃Cutter
P14：
P15：

Explanation：cutter holder clockwise/counterclockwise rotary is a single power point output（OC door）, current limit is 0.5A，it needs additional diode to retain current if it has external adductive load(such as direct current relay).

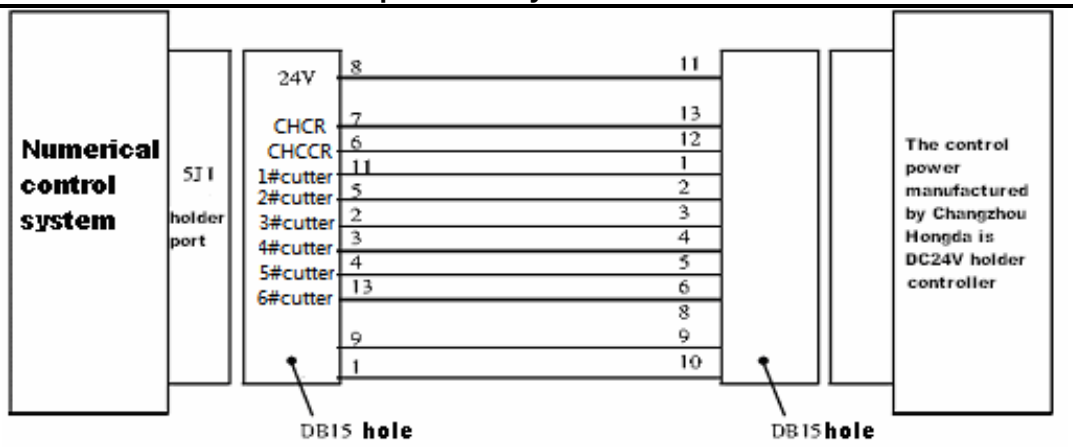
5J2 port provides internal current retain mode, if you connect P14 pin（internal current retain） to the power pin（such as P8，P15，+24V） that supplying power to system M function can realize current retain function, but it is forbidden to connect to 0V or earth. If the external adductive load (such as direct current relay) has been connected to a current retain diode, the connection to P14 pin is not necessary (and the system suggest to connect the external adductive load parallel connect to current retain diode).

1＃ cutter、2＃cutter、3＃cutter、4＃cutter、5＃cutter、6＃cutter、7＃cutter、8＃cutter are standing for different cutter position input of electric tool holder.

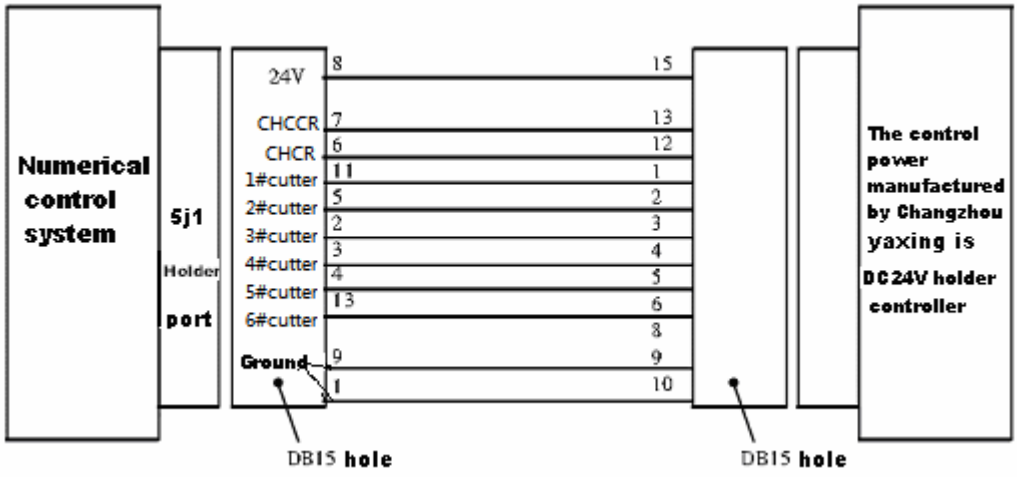
When the system is using external power supply, see the external connection principle diagram in Illus. 7-26(take four stations as example, if it is six stations, two additional wire must be connected to the 4#,13# pin which are on the side of system.).



When using system power supply, see the external connection in Illus. 7-27（take six stations as example, if it is four stations tool holder, the P4、P13 connection of CNC5J1 are not needed.）.



(a)



(b)

Illus. 7-27

If you are using other mode of cutter holding controller, please refer the connection accordingly to the manual

6.4.5 Motor port 4J1、4J3

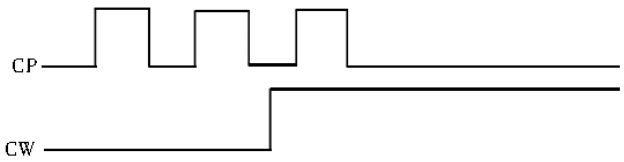
The motor signal port of 4J1、4J3 are the socket of 'DB15 Pin', the pin should be'DB15 socket', they separately output drive signal to X and Z axle motor. Port definition:（take X axle as example X，change the Z into Y, Z）

P1：GND P2：XEN P3：XTRF P4：XPSN P5：XERR

P6：XCP- P7：XCW P8：+5V P9：GND P10：XEN2

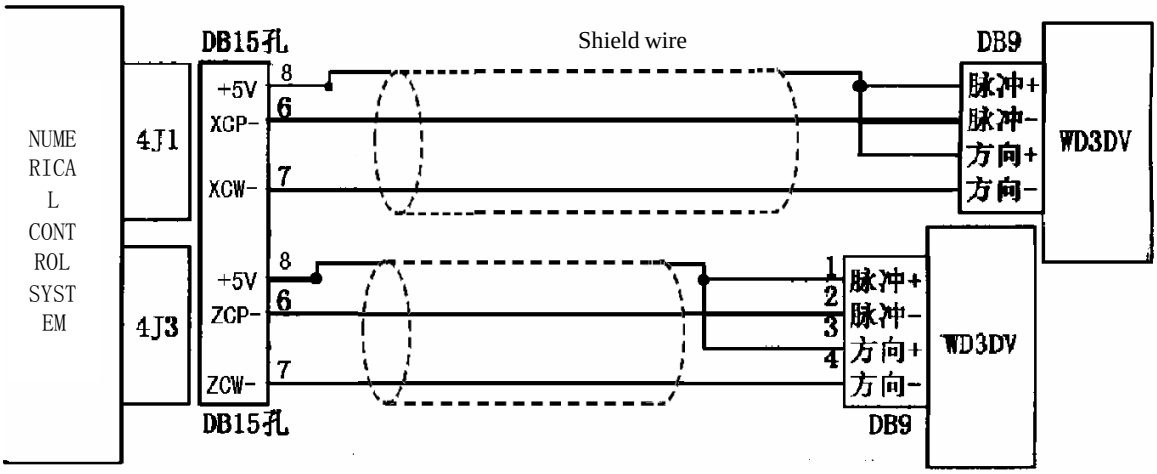
P11：XREF P12：XRDY P13：XCP+ P14：XCW+ P15：24V

The motor signal port only adopts hardware ring-sectioned stepping motor driver or pulse A/C servo unit. The output control signal mode：CP pulse and CW direction signal. CP outputs at positive pulse, and the corresponding motor moving forward one step according to one pulse. CW controls the motor to clockwise at high level, and counterclockwise at low level. The pulse signal is 1/2 duty，when the direction signal switch direction, the advance pulse is 1/4 duty.



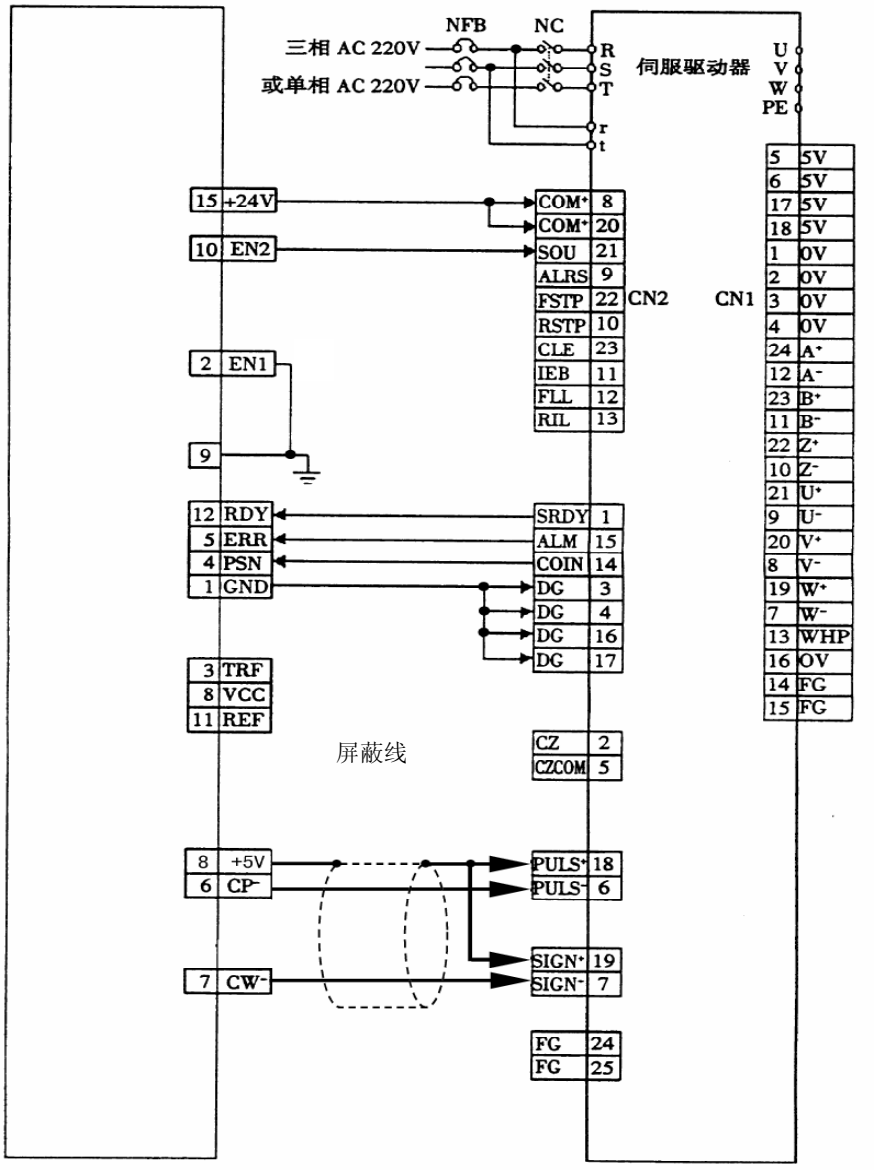
F6-28

The motor signal port is connected with triple-phase composite driver of our company as Figure:



F6-29 Connection diagram of motor signal port and triple-phase composite driver of our company

The connection of motor signal port and AC servo driver of our company is as Figure 6-30



F6-30 Motor signal port connection between A/C servo driver of WASHING CNC

Description: when the shielding layer is used to connect +5V or 0V ground terminal at both sides, the core wire in the shielding layer can't be used to connect +5V or 0V ground and more.

Other signal related with motor control, take X-axis for instance:

P2/P10 XEN1/XEN2: relay output contact pair, servo enable signal, inform servo to operate with power

on.

P12: XRDY: input, after the servo-unit accepts XEN1/2, self-check and lock with power on, send XRDY signal to CNC normally.

P3: XTRF: output, the servo zero finding signal (optional) forces servo to enter reference point returning state, after the servo-unit finds the X signal of encoder, stop locking immediately, meanwhile, return CNC XPSN signal.

P4: XPSN: input, when CNC stops at Z pulse position or eliminate the following error to set value, the CNC XPSN signal will be answered.

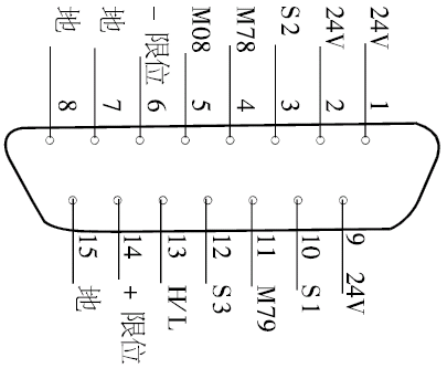
P5: XERR: input, when the servo-unit has error owing to certain cause or can't operate, this signal will be fed back to CNC.

P11: XREF: the servo-unit zero-returning (machine tool returns to reference point) signal can also connect the Z signal of the electric machine encoder to the XREF terminal, CNC will directly detect the Z pulse of machine tool and the zero of machine tool will be determined; XTRF is invalid, it's suggested that the manufacturer should adopt this method to return reference point.。

6.4.6 Input/output port 5J2

Input/output port 5J2 is‘DB15 pin ‘socket， the pin should be‘DB15 pin’. The port has 6 ways of relay power drive output signal and 3 ways of input signal. To input signal, we suggest when using external contact point switch, proximity switch(or Hall device), the device is at high level if it does not send out signal and it is at low level while it does send out signal, the driving capacity output at lower level is larger than 15mA, and it should adopt the device with power range DC10～30V.

P1：24V
P2：24V
P3：S2
P4：M78
P5：M08
P6：-Limit
P7：GND
P8：GND
P9：24V
P10：S1
P11：M79
P12：S3
P13：H/L
P14：+LIMIT
P15：GND



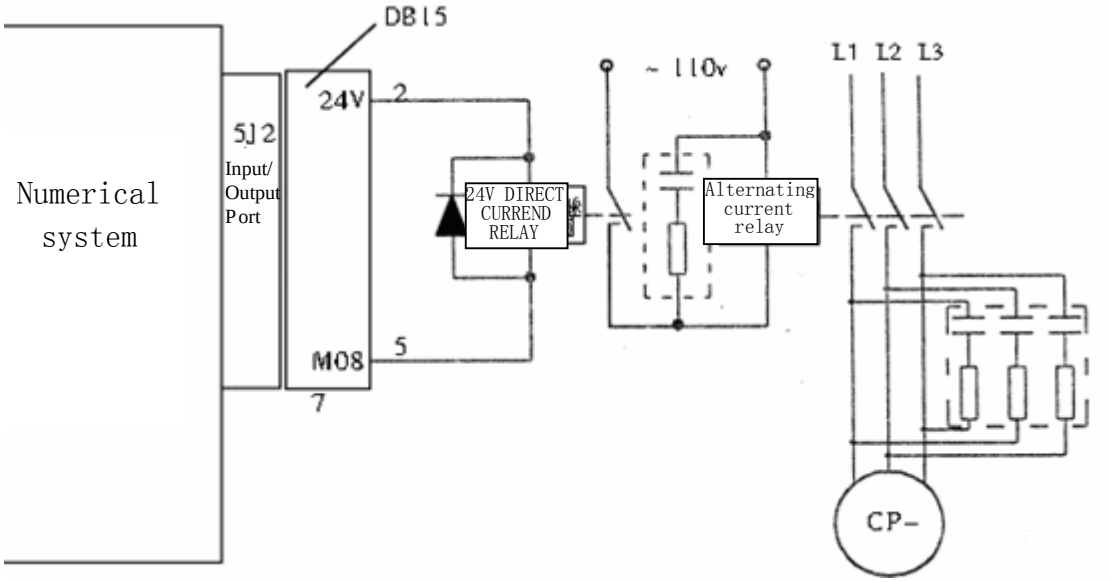
Explanation：S1，S2，S3，M78，M79，M08 are single power point output（OC door），limit current is 0.5A，it needs retain diode when with external adductive load（such as DC relay, etc）

S1，S2，S3：three speed motor output

M08：cool output

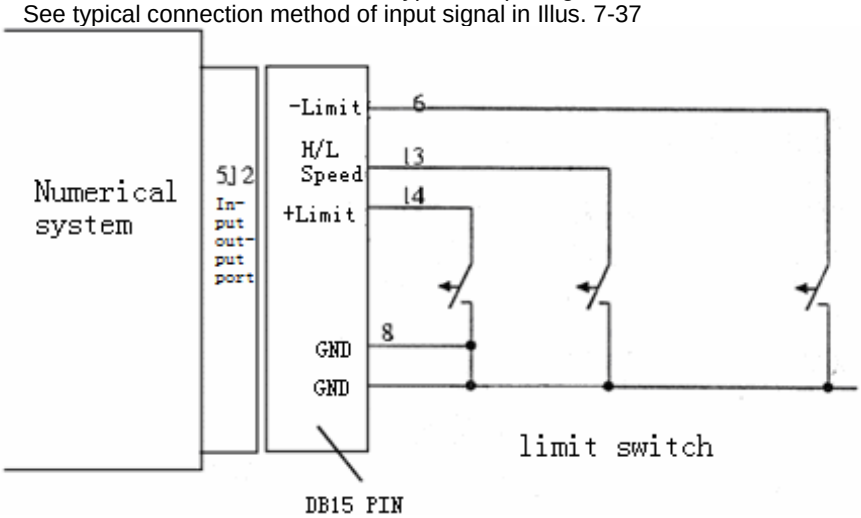
H/L：main axle high/low speed input

eel typical connection method of output signal in Illus. 7-36（take M08 as example）：

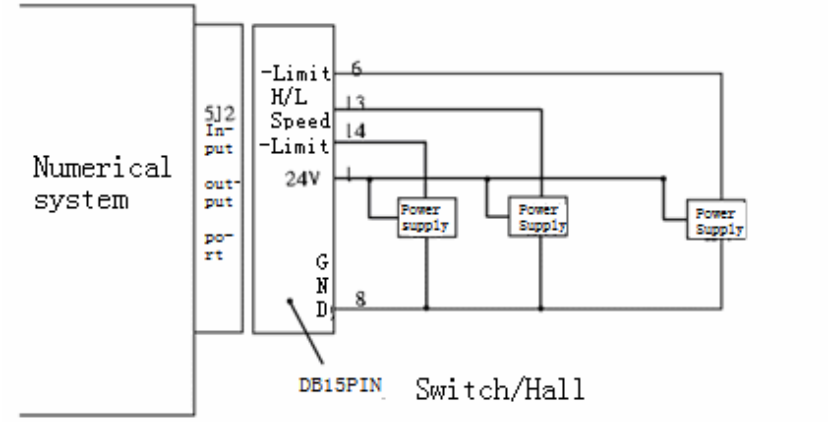


Illus. 7-36 Typical output signal connection method

See typical connection method of input signal in Illus. 7-37



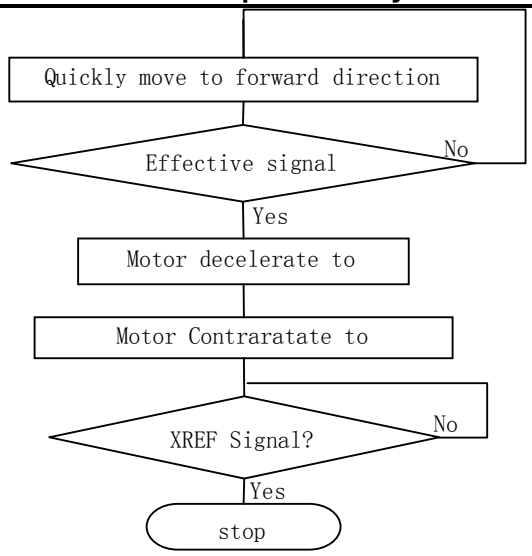
(a) The typical input signal connection method when using limit switch



(b) Typical input signal connection method when using power supply(such as proximity switch) device

Illus. 7-37 typical input signal connection method

Motion sequence of return mechanical zero point（machine tool zero point）:



Illus. 7-38

6.4.7 Spindle coder port 6J1

Spindle coder port 6J1 is'DB15 hole' mode socket，the connection pin is'DB15 pin'， This interface also includes a standby hand wheel interface (designed to be compatible with old system), user shall use standard '5J3 hand wheel interface' when connecting hand wheel:

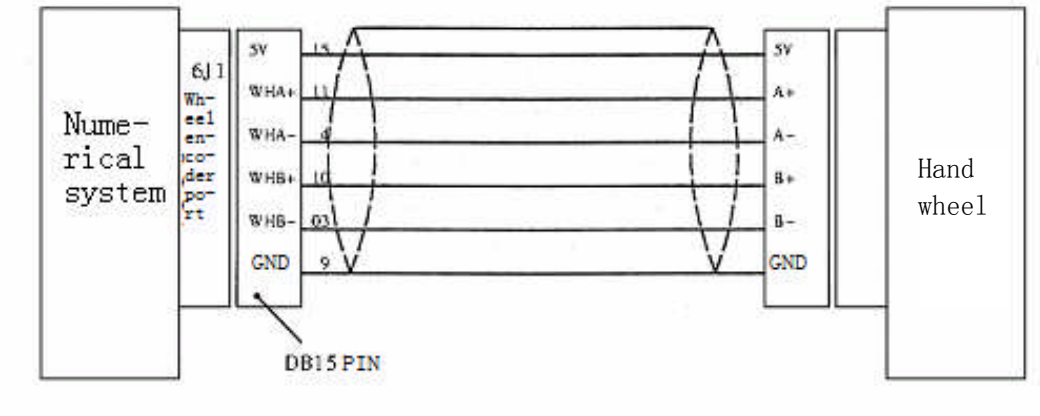
Pin	Function define	Pin	Function define
01	ground	09	+24V
02	ground	10	WHB+
03	WHB-	11	WHA+
04	WHA-	12	ENC+
05	ENC-	13	ENB+
06	ENB-	14	ENA+
07	ENA-	15	+5V
08	+5V		

1. working voltage：5V
2. pulse number per circle：100
3. output signal：two ways of difference output，i.e. A＋、A－ and B＋、B－

The coder connection with WA320 should meets the following condition：

1. working voltage：5V
2. pulse number per circle：700～2400
3. output signal：three ways of difference output，i.e. A＋、A－，B＋、B－ and Z＋、Z－

Hand wheel connection must use shield wire, and dual cored shield wire is more preferred. Two dual cored wire connect one difference signal, see method in Illus. 7-39.

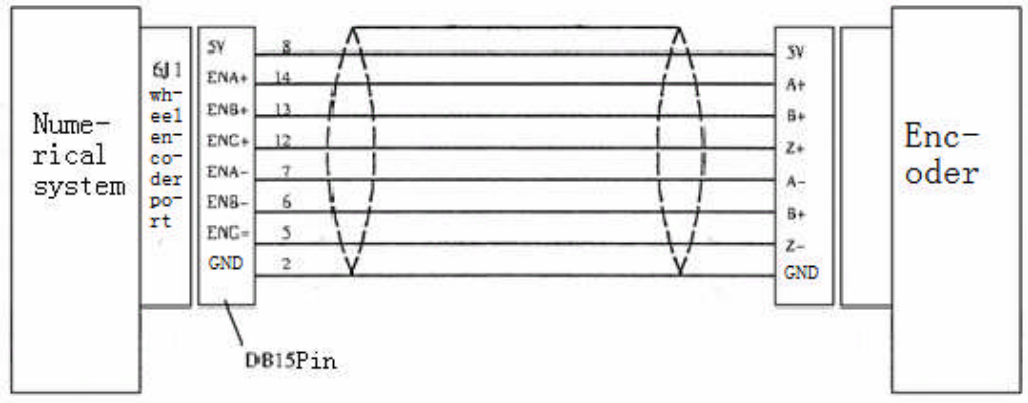


Illus. 7-39 hand wheel connection diagram

When hand wheel rotary direction reverse to CNC system defined coordinate, you should switch signal A＋ and B＋、signal A－ and B－ differently.

The coder connection must use shield wire, and dual cored shield wire is more preferred, two shield wires connect one way difference signal, the connection method in following diagram.

Explanation：Using the shield layer as ＋5V earth connection, and core wire is forbidden for ＋5V earth connection.



6.4.8 External hand wheel interface 6J2

The hand wheel and encoder interface 6J2 is 'DB15 hole' type socket, and the connected plug is 'DB15 pin', as figure:

Pin	Function define	Pin	Function define
01		09	Z axis
02		10	B+
03	B-	11	A+
04	A-	12	X axis
05	Ratio 10	13	Ratio 100
06	emergency stop	14	
07	+5V	15	+24V
08	GND		

System supports multiple hand-operated pulse generator access modes, and can freely choose current hand-operated pulse generator and parameters related with hand-operated pulse generator:

B121:=1: support external input multiplying factor and axis selection

=0: not support external axis selection and multiplying factor, which is selected by keyboard of system

B122:=1: current hand-operated pulse generator is selected by three-position switch

Three-position switch: start: hand-operated pulse generator connected with 6J1

Spindle stop: hand-operated control box connected with 6J2

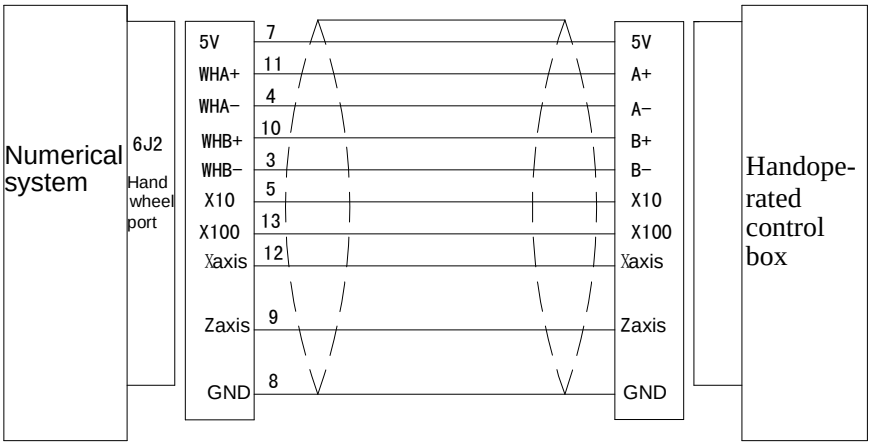
Feed stop: hand-operated pulse generator

=0: hand-operated pulse generator is selected by 115# parameter

Only adapted to the hand wheel which meets the following conditions (also called hand-operated pulse generator):

1. Operating voltage: 5V 2. Pulse number of each rotation: 100 3. Output signal: two-way differential output, namely A+, A- and B+, B-**Description:** O20 (output standby) belongs to single power point output (OC gate), limit current is 0.5A, and external reactive load (such as DC relay etc.) requires increasing fly-wheel diode.

Shielding wire must be used for hand wheel connection, and multiple twin shielding wire shall be used as possible, two twisted-pairs are connected with one-way differential signal as Figure below:

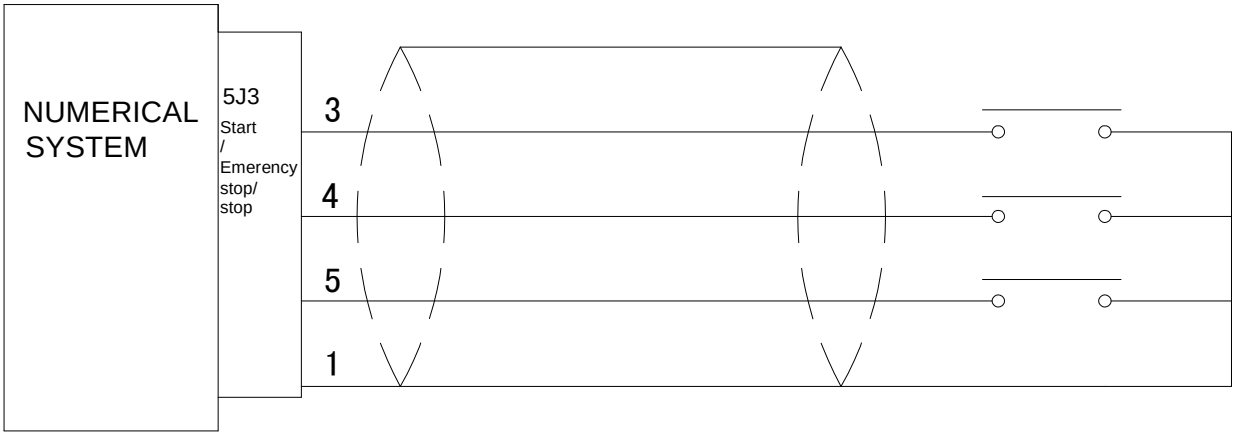


6.4.9 External Start emergency stop and pause interface 5J5

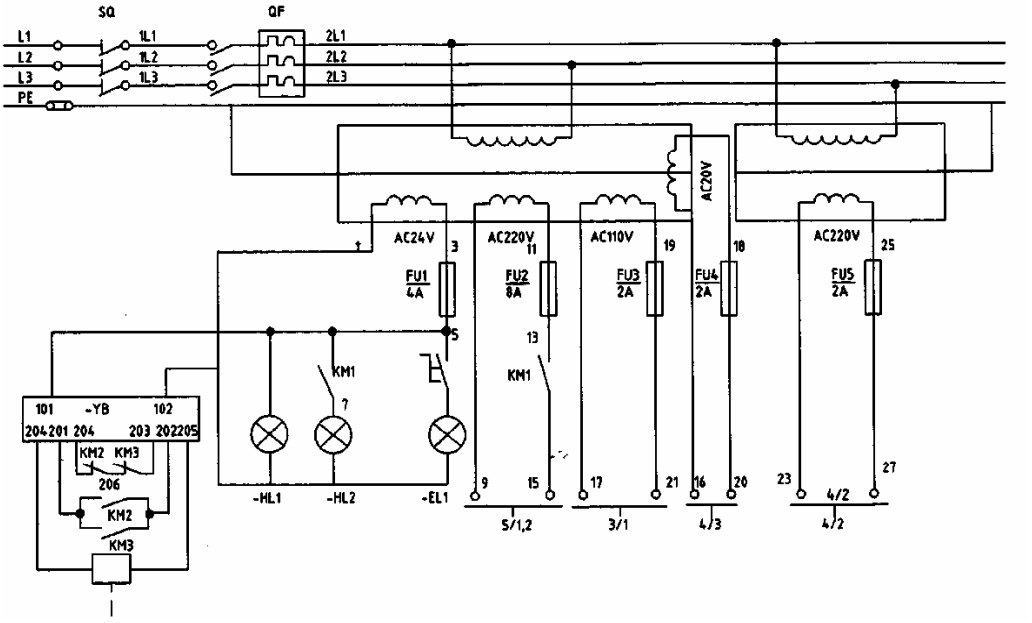
The model of this interface is 'DB9 port', and the corresponding plug shall be 'DB9 pin'. Definition is as follows:

Pin	Function define	Pin	Function define
01	GND	06	GND
02	GND	07	Start
03	Start	08	Pause
04	Pause	09	emergency stop
05	emergency stop		

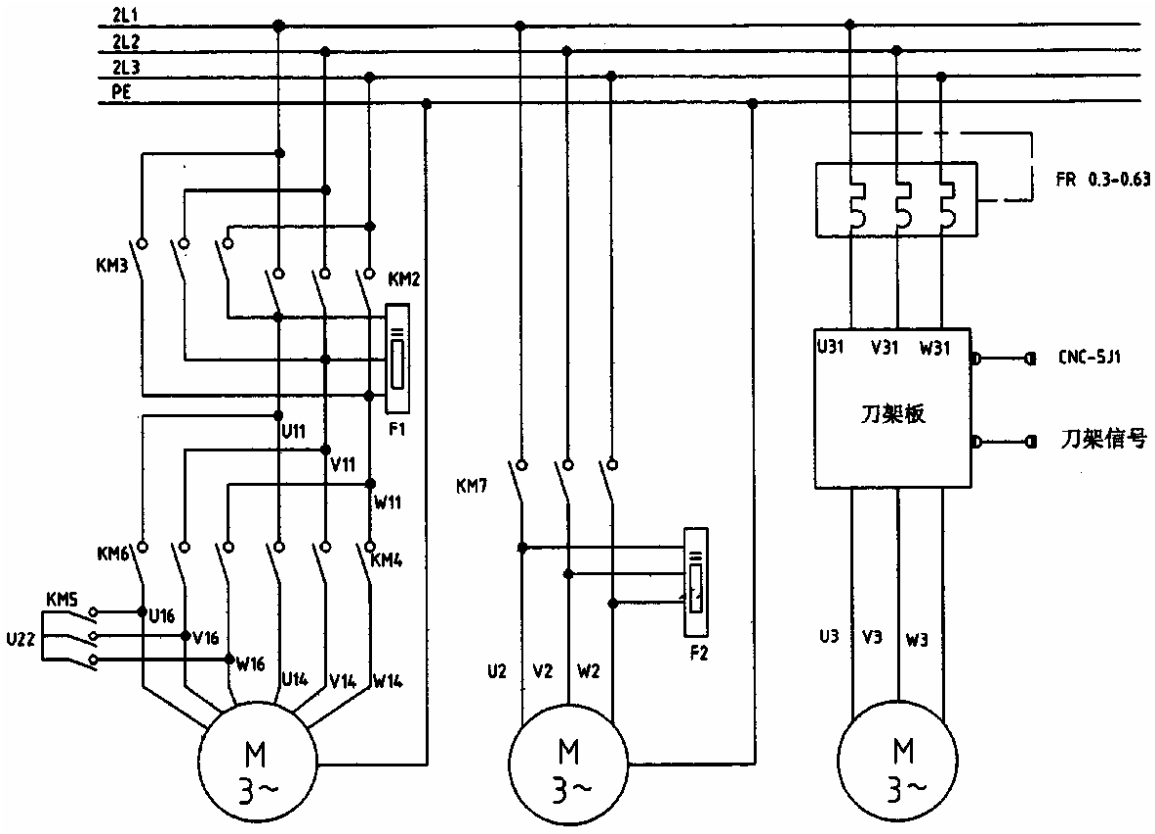
As follows:



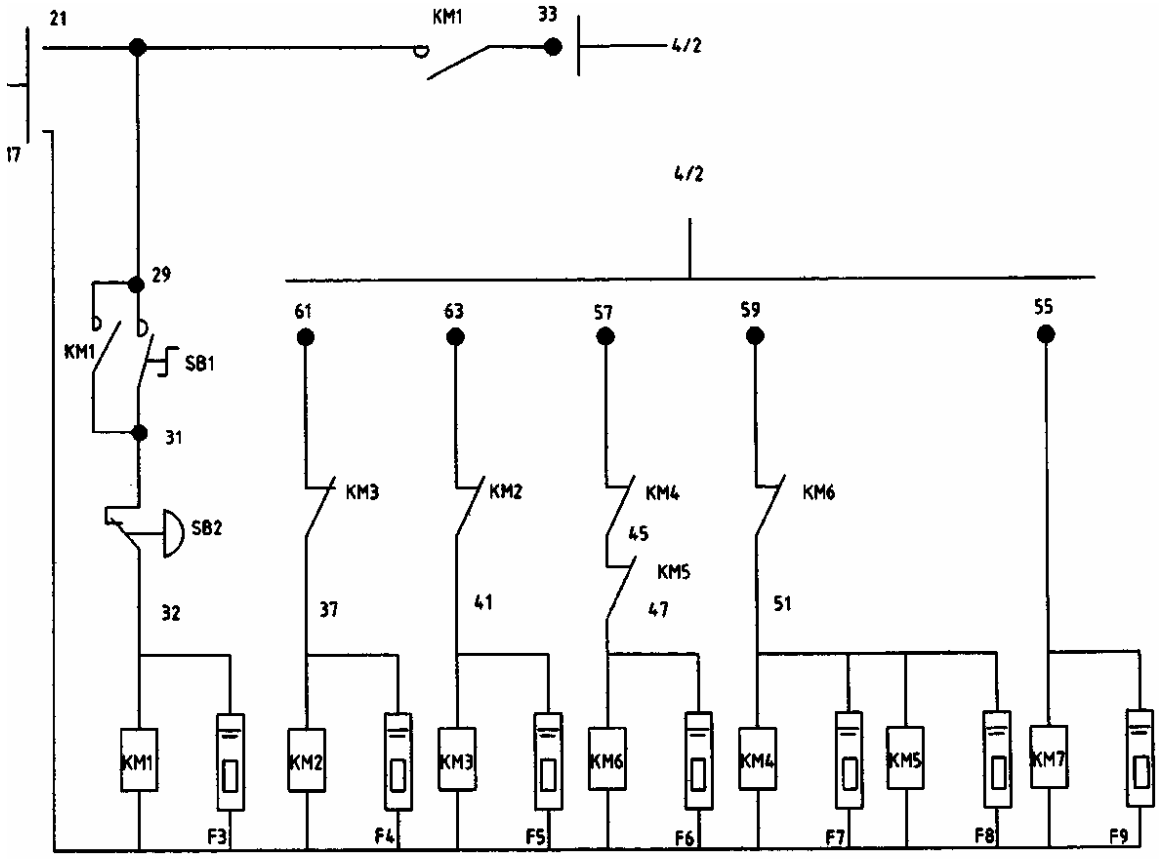
6.5 Typical electric application scheme



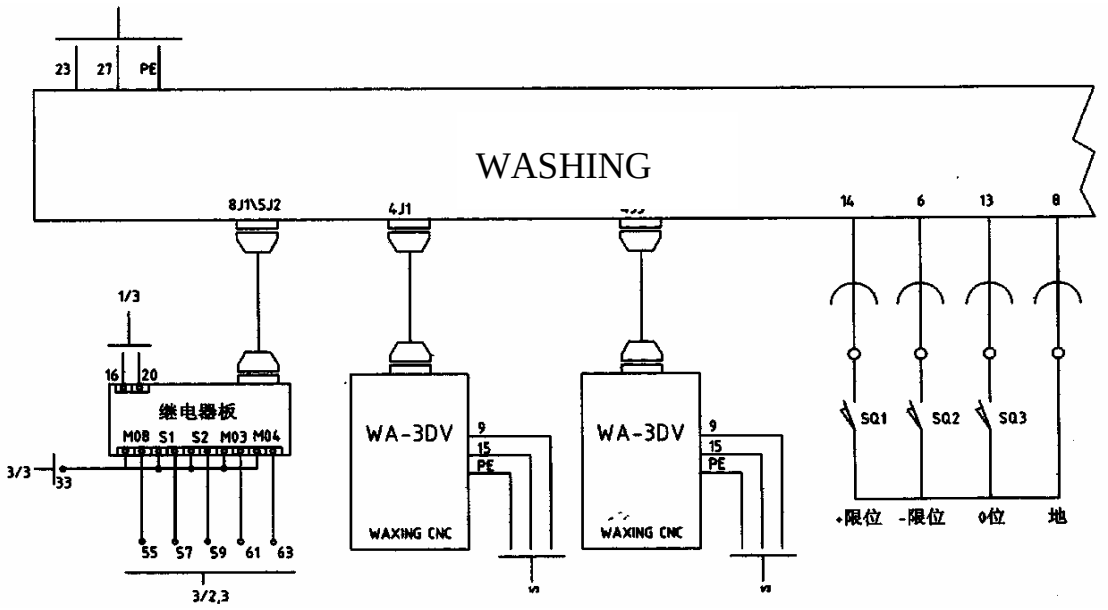
F6-37



F6-38


21
KM1
33
4/2
17
4/2
29
KM1
SB1
31
SB2
32
61
63
57
59
55
KM3
KM2
KM4
KM6
45
KM5
37
41
47
51
KM1
KM2
KM3
KM6
KM4
KM5
KM7
F3
F4
F5
F6
F7
F8
F9

F6-39


23
27
PE
WASHING
8J1\5J2
4J1
14
6
13
8
1/3
16
20
继电器板
M08 S1 S2 M03 M04
3/3
33
55 57 59 61 63
3/2,3
WA-3DV
WAXING CNC
9
15
PE
WA-3DV
WAXING CNC
9
15
PE
SQ1
SQ2
SQ3
+限位
-限位
0位
地

F6-40

Appendix One Error alarm

Notes of error number and error content:

Error number Error content

01 The time defined by G04 has error

02 Undefined K parameter

03 Error of G24 sub-program returning and disorder of transition processing and sub-program call

04 Data exceeds after G31 magnifies

05 Undefined

06 The cutting tool number or cutting tool compensation number has errors when starting up.

07 Without the G and M functions

08 Transition processing notification has error (including sub-program call)

09 Undefined

10 The first letter of program line has error (the line must be initiated with N)

11 Undefined

12 CNC doesn't memorize work piece coordinate before calculating cutting tool compensation value, using Xsav or Zsav keys.

13 The data format has errors such as numerical value after coordinate, the cutting tool compensation value which is correspondent to cutting tool number (such as T01) in the cutting tool parameter table, numerical value in the P parameter, four digits are required before decimal and three digits after decimal.

14 Transition processing doesn't find completion segment number

15 There are too many characters or illegal characters appear in the line

16 Tool compensation has errors

17 Cutter compensation plane fault

18 Undefined

19 The establishment of cutter compensation did not use catch-G01

20 There have been M02 and M30 when compiling M00

21 What's called by G20 is not sub-program，C cutter compensation processing Parks radius of the arc to zero or negative

22 Undefined

23 The thread pitch is too big or too small or thread lacks Z and K

24 The object segment number of cycle processing has error or is not found

25 Undefined

26 Transition processing shouldn't appear in the last line, and should add M02

27 The arc lacks parameter of Cutter compensation or fill plane does not coordinate with the compensation

28 Undefined

29 Zero length of cutter compensation track

30 Processing can not operate this function

31 232 Serial communications error

32 File is not found or file has error

33 Undefined

34 Memory is full

35 Undefined

36 Undefined

37 Undefined

38 Undefined

39 Two parameters after G and M have errors

40 Limit

41 Drive alarm

42 General alarm

43 Undefined

44 The time of holder reversion is too long and there is no tool position signal

45 Definition of filename has error, or the communication has error when implementing input/output

46 Undefined

47 Undefined

48 This file can't be used owing to disorder of the file internal addresses

49 Files show that the wrong character

50 The arc start point doesn't conform to end point

51 The whole circle can't be programmed with R

52 Three-position switch isn't at start position when processing is started

53 When the optional segment starts, the corresponding segment number is not found

54 The angle-specified stop of coordinate axis motor isn't realized

55 Emergency stop alarm

56 Feed speed is zero when feeding of each rotation。

57 Undefined

58 DNC memory is too small。

59 Spindle start-up is error。

66 The cutter number isn't checked when the mechanical zero isn't returned or zero is returned after starting up.

67 Mechanical zero signal is error.。

68 The spindle clamping ready signal isn't checked

71 The key of keyboard is pressed or the initiate key isn't ejected when starting up, it'll start wrongly。

72 Thread acceleration is 0

73 The length of thread is too short and can't be processed (Z-direction acceleration can be enhanced)

74 When thread processing is performed, spindle speed isn't stable or internal data format is wrong

75 Encoder feedback is 0 or Z-direction acceleration is too high when thread processing is performed

78 Circular interpolation has error

80 Without this I/O port

82 Internal processing data format is wrong/power down protection data is wrong

83 Illegal processing function

84 Write error of electronic disk

85 Memory is insufficient when 232 communication is performed

86 Read error of electronic disk

87 Contact with upper computer has error when communication is performed

88 232 Serial communications error

89 Same as 88#

93 USB error

94 Servo-shaft un-enabled

95 U disk can't be used during processing

97 C cutter compensation track has no cross point

98 Trial period is set wrongly

99 Trial period is expired

Appendix Two System parameter

Parameter Number	Scope	Factory set	Recommend scope
00	G00 Fast locating speed setting (mm/min)	200	500-6000
01	Holder reverse lock time set (second)	0.8	0.1-10
02	The cutting tool number of machine tool	4	1-10
03	The spindle 1 gear rotating speed top limit (rpm) (used for frequency converter analogue output)	3000	100-8000
04	The spindle 2 gear rotating speed top limit (rpm) (used for frequency converter analogue output)	1000	100-8000
05	＝0 Machine tool of electro-holder; ＝1machine tool array and change	1	0-10
06	X axis direction clearance (1 min)	0	0-1.0
07	Z axis direction clearance (mm)	0	0-1.0
08	The coordinate of X-axis presetting cutter point which is relative to machine tool zero	0	0-±99999
09	The coordinate of Z-axis presetting cutter point which is relative to machine tool zero	0	0-±99999
10	Manual fast speed (mm/min)	5000	0-15000
11	Spindle reversing time delay (second)	0.2	0-65
12	Spindle braking time delay (second)	0	0-65
13	Spindle starting time delay (second)	0.5	0-65
14	M05 relay pulse output time delay (second)	0	0-65
15	Relay pulse output time delay (M71—M85)(second)	1.0	0-65
16	Screw in and screw out speed in the thread X direction (see G86 description for detailed information)	5000	0-10000
17	Initial image, ＝0 version;＝1 manufacturer description (option); ＝2 operating image	0	0-2
18	X-axis presetting cutter point set value which is relative to work piece coordinate	0	0-±99999
19	X-axis presetting cutter point set value which is relative to work piece coordinate	0	0-±99999
20	Spindle pulse number per rotation	1200	700-4800
21	M04 output =0: maintaining output; ＝1: pulse output	0	0-±99999
22	＝0：M05 close S1、S2、S3； =1：M05 doesn't close S1、S2、S3	0	0-±99999
23	Percentage of unstable spindle speed, the thread can be processed when it's less than this value	5	5-20
24	Undetermined	0	0-±99999
25	The smoothing tool amount of the last cutting when processing thread,=0: without smoothing tool (mm)0	0	0-65
26	Undetermined	1000	1-5000
27	The segment number increments 0—99 are automatically generated when editing program; the segment number is not generated when it's equal to 0	10	0-100

28	The low limit of spindle rotating speed in the time of constant linear speed cutting.	100	1-5000
29	The brightness value of LCD when starting up (10—32)	12	0-32
30	The time constant of X direction when it's G00 (millisecond)	6000	500-60000
31	The time constant of Y direction when it's G00 (millisecond)	6000	500-60000
32	The time constant of Z direction when it's G00 (millisecond)	6000	500-60000
33	The electronic gear multiply factor of X direction	1	0-1000
34	The electronic gear percentage of X direction	2	0-1000
35	The electronic gear multiply factor of Y direction	1	0-1000
36	The electronic gear percentage of Y direction	1	0-1000
37	The electronic gear multiply factor of Z direction	1	0-1000
38	The electronic gear percentage of Z direction	1	0-1000
39	The time constant of various axes when cutting operates (millisecond)	100	50-1000
40	The time constant of Z axis when processing thread (millisecond)	250	50-1000
41	Speed top limit when Cutting	5000	200-15000
42	Speed low limit when G00 is fast	500	500-15000
43	Looking for slow speed of zero signal when returning reference point	50	1-500
44	The time constant of each axis of soft limit to stop	200	50-1000
45	Speed top limit for thread cutting (Z direction)	5000	2000-15000
46	Input signal de-jitter number	12	3-30
47	Time delay between electro-holder positive and reverse rotation	0.05	0.0-10
48	Speed top limit of clearance compensation (time constant is same as 39#)	2000	1000-10000
49	The time constant of X direction screw in and screw out de-trailing when processing thread	250	50-1000
50	Reverse clearance value of Y axis	0	0-1.0
51	The time constant for various axes of hand wheel operating	100	100-1000
52	The interval length for thread pitch error compensation of X-axis	0	0-60
53	The point number of X-axis thread pitch error compensation	0	0-160
54	The interval length for thread pitch error compensation of Y-axis	0	0-60
55	The point number of Y-axis thread pitch error compensation	0	0-160
56	The interval length for thread pitch error compensation of Z-axis	0	0-60
57	The point number of Z-axis thread pitch error compensation	0	0-160
58	The limit for contour error of circular interpolation (min)(0.002 in general)	0.002	0.001-0.01
59	The screw in/de-trailing speed top limit of X direction when processing thread	5000	100-15000
60	The positive direction soft limit coordinate of X-axis from the reference point	0	0-99999

61	The negative direction soft limit coordinate of X-axis from the reference point	0	0-99999
62	The positive direction soft limit coordinate of Y-axis from the reference point	0	0-99999
63	The negative direction soft limit coordinate of Y-axis from the reference point	0	0-99999
64	The positive direction soft limit coordinate of Z-axis from the reference point	0	0-99999
65	The negative direction soft limit coordinate of Z-axis from the reference point	0	0-99999
66	The input terminal number of X-axis reference point coarse positioning signal	27	1-40
67	The input terminal number of Y-axis reference point coarse positioning signal	0	1-40
68	The input terminal number of Z-axis reference point coarse positioning signal	11	1-40
69	The input terminal number of X-axis reference point fine positioning signal	26	1-40
70	The input terminal number of Y-axis reference point fine positioning signal	0	1-40
71	The input terminal number of Z-axis reference point fine positioning signal	28	1-40
72	Positive direction limit input terminal number	3	1-40
73	Negative direction limit input terminal number	4	1-40
74	External general alarm input terminal number	0	1-40
75	Drive alarm input terminal number	0	0-±99999
76	Spindle high/low speed input terminal number	5	1-40
77	The spindle 3 gear rotating speed top limit (rpm) (used for frequency converter analogue output)	1000	100-8000
78	The spindle 4 gear rotating speed top limit (rpm) (used for frequency converter analogue output)	1000	100-8000
79	The top limit for holder positive rotation (44# alarm occurs at the start of holder positive rotation)	8	1-8
80	Position ring constant m (0—50)	0	
81	Position ring constant KP (5—100)	80	
82	Position ring constant KD (0—50)	0	
83	The spindle reversion time delay at the time of rigidity threading	0	0-65
84	Self-specified thread tooth type angle I, used for single side thread cutting.	0	
85	Self-specified thread tooth type angle II, used for single side thread cutting.	0	
86	Speed for each axis of returning to machine tool zero	6000	0-15000
87	The feed length increased of each millimeter at the time of rigidity threading	0	0-15
88	The feed speed of thread X direction	3000	1000-5000

89	The waiting time of feed axis angle-specified stop (millisecond)	20.000	0-65
90	X-axis driving alarm input port number	16	1-40
91	Y-axis driving alarm input port number	0	1-40
92	Z-axis driving alarm input port number	1	1-40
93	A-axis driving alarm input port number	6	1-40
94	Spindle start-up normal input port number	0	1-40
95	Spindle clamping solenoid valve output port number	12	1-16
96	Spindle loosing solenoid valve output port number	13	1-16
97	Spindle clamping pedal swith input port number	10	1-40
98	Spindle loosing pedal swith input port number	12	1-40
99	Servo open high voltage main circuit output port number	0	1-16
100	Time delay from system power-on to servo open high voltage (second)	0	0-65
101	Time delay from opening servo high voltage to output servo enabled (second)	0	0-65
102	Three-position switch loop start bit input port number	30	1-40
103	Three-position switch spindle stop input port number	31	1-40
104	Time delay from outputing servo enabled to open servo-motor internal contracting brake (second)	0	0-65
105	Spindle hydraulic clamping oil cylinder power-on time (second)	0	0-65
106	Spindle hydraulic loosing oil cylinder power-on time (second)	0	0-65
107	Spindle hydraulic clamping in-position signal input port number	0	1-40
108	Time of checking spindle hydraulic clamping (second)	0	0-65
109	Lubricate output port number in fixed time	0	1-16
110	Interval of lubricating in fixed time (second)	0	0-99999
111	Start-up time of lubricating in fixed time (second)	0	0-3600
112	External cycle pause switch input port number	7	1-40
113	Power-on open servo R S T strong power relay delay (second)	0	0-65
114	Set value of Y-axis preset cutter point corresponding to the workpiece coordinate (mm)	0	0-±99999
120	Operation machining default display mode	1	0～2
121	Tailstock forward output port number	0	1～16
122	Tailstock backward output port number	0	1～16
130	86#=0, X-axis reference point return speed	0	0～60000
131	86#=0, Y-axis reference point return speed	0	0～60000
132	86#=0, Z-axis reference point return speed	0	0～60000
133	86#=0, A-axis reference point return speed	0	0～60000
146	Machining piece number set, after the piece reaches the set value, system will give alarm to stop machining	0	
149	Set rotation speed for normal, and the testing time is 13#	0	

Appendix Three **Digit parameter**

00#

B001	B002	B003	B004		B006	B007	B008

B001： =1： Radius programming =0： Diameter programming

B002： =1：The function of milling machine interface =0：The function of lathe interface

B003： =1：When the program execution completes, M05 and M09 are not inserted automatically

=0：When program execution completes, M05 and M09 are inserted automatically

B004：=1：the third movement axis are opened under lathe execution mode，whose function is valid when B001=0.

B006：=1：Z axis opens electronic gear function = 0 ：Z axis doesn't open electronic gear function.

B007： =1：Y axis opens electronic gear function =0：Y axis doesn't open electronic gear function.

B008： =1：X axis opens electronic gear function =0：X axis doesn't open electronic gear function.

00# factory set is 0000011

01#

B011	B012	B013	B014		B016	B017	B018

B011 ： =1 ： Dynamic thread pitch error compensation values are displayed on the operation interface，and their values are respectively signified with XH、YH and ZH

B012： =1：M03，M04 is pulse output

=0：M03，M04 is long signal output

B013： =1：M05 closes S1～S4

=0：M05 doesn't close S1～S4

B014： =1： Recover the rotation of previous spindle after threading ends

=0： Not to recover the rotation of previous spindle after threading ends

B016： =1：Z axis opens thread pitch error compensation function

B017： =1：Y axis opens thread pitch error compensation function

B018： =1：X axis opens thread pitch error compensation function

01# factory set is 00000000

02#

B021	B022	B023	B024		B026	B027	B028

B021： =1：The reversion clearance compensation condition is displayed dynamically on the operating interface, and signified with XK 、YK 、ZK

B022： =1：Open soft limit function

B023： =1：Each axis stops suddenly at the soft limit

=0：Each axis stops with speed reduction（recommendation）

B024： =1： It's valid that the soft limit mustn't return to the reference point.=0：It's valid that the soft limit must return to the reference point

B026： =1：Z axis opens reversion clearance compensation

B027： =1：Y axis opens reversion clearance compensation

B028： =1：X axis opens reversion clearance compensation

02# factory setting is 00000111

03#

B031	B032	B033			B036	B037	B038

B031：=1：Self-verifying is passed after starting up, and CNC outputs ENABLE signal to the servo driver （each axis and contact point output）

= 0 ： self-verifying is passed after starting up, and CNC doesn't output ENABLE sign to the servo driver （each axis and contact point output）

B032： =1：When G00 fast moves, the speed modification and regulation is valid but it cannot exceed the top limit of fast speed which is determined by the 00# Parameter.

=0 ： G00 zero speed modification and regulation.

B033: =0: The auto-cycle can be valid that the system must return to reference point (machine tool zero) after power-on.

=1: The auto-cycle can be valid that the system doesn't require returning to reference point (machine tool zero) after power-on

B034: =0: Recover the work piece coordinate of current cutter after returning to the reference point of machine tool (REF=0 must be valid)

=1: The work piece coordinate is reset to zero after returning to the reference point of machine tool.

B036： =1：The function of Z axis returning to the reference point is opened,

=0：The function of Z axis returning to the reference point is unopened.

B037: =1：The function of Y axis returning to the reference point is opened,

=0：The function of Y axis returning to the reference point is unopened.

B038： =1：The function of X axis returning to the reference point is opened,

=0：The function of X axis returning to the reference point is unopened.

03# factory set is 10000111

04#

B041	B042	B043	B044	B045	B046	B047	B048

B041-B048 respectively indicate whether the input level from 8# output port to 1# output port is high level (=1) validity or low level (=0) validity

04# factory set is 00000000

05#

B051	B052	B053	B054	B055	B056	B057	B058

B051-B058 respectively indicate whether the input level from 16# output port to 9# output port is high level (=1) validity or low level (=0) validity

05# factory set is 00000000

06#

B061	B062	B063	B064	B065	B066	B067	B068

B061-B068 respectively indicate whether the input level from 24# output port to 17# output port is

high level (=1) validity or low level (=0) validity

06# factory set is 00000000

07#

B071	B072	B073	B074	B075	B076	B077	B078

B071-B078 respectively indicate whether the input level from 32# output port to 25# output port is high level (=1) validity or low level (=0) validity

07# factory set is 00000101

08#

B081	B082		B084		B086	B087	B088

B081：=1：spindle M function is output with 4-gear speed rotation and the 4-gear s peed is selected by M41-44, which is suitable for converter with mechanical gear of the 4 gear. The 4-gear speed is determined by the parameters of 3#、4# and 77#.

=0： Select spindle high and low signal, which is suitable for the spindle converter with high and low speed mechanical conversion, and high and low signal is input into the system. The analogue top limit for 3# or 4# of P parameter is determined by the spindle high and low signal.

B082: =1: Select spindle analog ±10V output mode (option, special order)

B084: =0: Output servo enabling signal after being powered on and waiting servo to be ready

=1: Output servo enabling signal after being powered on and then wait for the servo ready signal

B086： =1：The reversion movement of X , =0：The positive movement of X

B087： =1：The reversion movement of Y , =0：The positive movement of Y

B088: =1：The reversion movement of Z , =0：The positive movement of Z

08# factory set is 00000000

09#

B091	B092				B096	B097	B098

B091：=1：Open flexible treatment of thread processing =0： It is not opened.

B092：=0：Soft limit is determined by the machine tool coordinates =1：The soft limit is determined by the work piece coordinates.

B096：=1：When the CNC is powered on, it will send ENABLE contact signal to servo output, later checking whether the READY signal of Z direction servo is received. If it's not received, the error of "Drive is not ready" is displayed on the operating interface.

=0: The READY signal of servo isn't checked.

B097：=1：When the CNC is powered on, it will send ENABLE contact signal to servo output, later checking whether the READY signal of Y direction servo is received. If it's not received, the error of "Drive is not ready" is displayed on the operating interface.

=0: The READY signal of servo isn't checked.

B098：=1： When the CNC is powered on, it will send ENABLE contact signal to servo output, later checking whether the READY signal of X direction servo is received. If it's not received, the error of "Drive is not ready" is displayed on the operating interface.

=0: The READY signal of servo isn't checked

09# factory set is 00000000

10#

B101	B102	B103	B104	B105	B106	B107	B108

B101： =1：Low vibration mode when the feed is selected

=0: Selecting high precision mode of speed

B102： =1：The S function is disallowed to implement S1, S2, S3 and S4 to control the spindle 4 gear electrical shift

=0: The S function is allowed to implement S1, S2, S3 and S4 to control the spindle 4 gear electrical shift.

B103：=1：The data reception speed is Ⅱ gear when DNC processes.

＝0: The data reception speed is I gear when DNC processes.（Recommendation）

B104： =1: When DNC starts, it first sends XON character to the upper machine tool

=0：DNC doesn't send XON character.

B105： =1：When serial communication is performed， The first line of numerical control program has no ％

=0：When serial communication is performed，， The numerical control begins with ％

B106： =1：The X direction is signified with circumference, 0—360 degree =0: The X direction is signified with length

B107： =1： The Y direction is signified with circumference, 0—360 degree =0: The Y direction is signified with length

B108： =1：The Z direction is signified with circumference, 0—360 degree =0: The Z direction is signified with length

The factory set of 10# is 000000

11#

B111	B112	B113			B116	B117	B118

B111： =1：Special machine tool control (the function is determined according to the special machine quality). =0: General system control.

B112：=1：Emergency stop/M77 is output at limit, (the alarm lamp can be controlled) =0: Don't output alarm signal.

B113：=1：The spindle analogue selects 0—5V. =0: The spindle analogue selects 0—10V.

B116：=1：One switch that X direction returns to reference point, it's the coarse positioning when pressing and fine positioning when releasing

=0: Two switches that X direction returns to reference point, and the coarse positioning and fine positioning are separated

B117：=1：One switch that Y direction returns to reference point, it's the coarse positioning when pressing and fine positioning when releasing

=0: Two switches that Y direction returns to reference point, and the coarse positioning and fine positioning are separated

B118：=1：One switch that Z direction returns to reference point, it's the coarse positioning when pressing and fine positioning when releasing

The factory set of 11# is 00000000

12#

B121	B122	B123	B124	B125	B126	B127	B128

B121:=1: External hand-operated pulse generator is effective, and the axis selection and multiplying factor is controlled externally

=0: Adopt keyboard to set axis selection and multiplying factor

B122: =0: Serial communications baud rate is 9600

=1：Baud rate is 19200 (pre-option, special order)

B123: =0: Chinese interface

=1: English interface

B124:=1: Timing lubrication starts. The lubrication interval is determined by 110#, the lubrication time is determined by 111# and the output port is determined by 109#.

B125:=0: When the file is typed in serially or DNC is transferred, the file header has no %

=1: Under the condition above, the file is started with %

B126:=0: G97 recovers the previous analog quantity after canceling the constant speed processing

=1：Not to recover

B127：=0： Spindle analog quantity outputs 0～5V or 0～10V.

=1： Spindle analog quantity outputs -10V～10V，0～-10V is reversion，0～10V is positive rotation.

(B127 is option which must be customized)

The factory set of 12# is 00000000

13#

B131	B132	B133	B134		B136	B137	B138

B131=0：The spindle S programming, it's valid with M03. =1: The spindle S programming, it's valid without M03.

B132=0：31 series system has no S4 function (only S1, S2 and S3) =1: 31 series system has S4 function.

B133=0：Defining M78 as S4 output (it's only valid when ES4=1) =1: Defining M79 as S4 output

B134：=0：M05 closes analog quantity output. =1：M05 doesn't close analog quantity output

B136：=0 Z-axis returns zero in positive direction =1 Z-axis returns zero in negative direction

B137：=0：Y-axis returns zero in positive direction =1 Y-axis returns zero in negative direction

B138：=0：X-axis returns zero in positive direction =1 X-axis returns zero in negative direction

The factory set of 13# is 10000000

14#

B141	B142	B143	B144	B145		B147	B148

B141:=0: External on-off signal detection spindle start-up

=1: When the spindle encoder checks that the spindle speed is greater than 50RPM, spindle will start normally

B142:=0: The general external alarm only gives prompt (error 42)

=1: The general external alarm is used as severe alarm, and the stop movement control is switched to manual operation

B143:=1: When the third axis (Y axis) of machine tool is effective as spindle position/speed mode, M05 sends M29 (switched to position mode)

=0: When the above is effective, M05 doesn't send M29 (maintain the speed mode)

B144: After M03 and M04 command start the spindle, they will check whether spindle starts normally

=1: Check whether spindle starts normally

=0: Not to check whether spindle starts normally

The testing time is determined by No. 11 of system, if No. 11 time is exceeded, 59# error will be given

B145: The third axis Y-axis working mode

=1: Y-axis is the main axis which can switch position/speed mode

=0: Y axis is the straight line position moving axis of normal axis

B147:=0: When the electric holder changes tool, confirm the tool number after completion.

=1: Not to confirm tool number after the tool is changed

B148： It's internal parameter which must be set for 0

The factory set of 14# is 00000000

15#

B151	B152	B153	B154	B155	B156	B157	B158

B151:=0: Cutter compensation modification value work piece coordinate

=1: Cutter compensation modification values minus work piece coordinate

B152:=1: The access of system three-position switch is valid =0: Invalid

B153:=1: Display the spindle actual measurement rotation speed on the operating interface, processing calculation and accumulate process time

=0: Not display

B154: The parity mode of tool number of special type holder (one of P5#=2～10 must be specified)

=1： Tool number odd parity

=0: Tool number even parity

B155：=0： Open keyboard buzzer

=1： Close keyboard buzzer

B156：=1： Cancel servo high-voltage relay output after driving alarm

=0： Not cancel

B157：=1： Cancel driving enable signal after driving alarm

=0: Not to cancel driving enable signal after driving alarm

B158：=0： Adopt various axes combined direction limit

=1： Limit of various axes and various directions independent parted mode (option)

The factory set of 15# is 00000000

16#

Option. Function option of pressing RESET key or soft reset is performed, to be updated. It's valid by selecting 1 (it's valid for software version V5.1 above)

RF8	RF7	RF6	RF5	RF4	RF3	RF2	RF1

RF1: Spindle

RF2：Cooling and lubrication

RF3：M31-M38

RF4：Switched to manual operation

RF5：Keep internally

RF6：Keep internally

RF7: Movement control ends

RF8：Keep internally

17#： Special machine tool category option:

The special functions such as water-jet flow, grinding machine laser and flame cutting can be respectively selected, please contact supplier for details

18#

B181		B183	B184		B186	B187	B188

B181: During the course of executing automatic processing program, whether M10 and M11 are

permitted valid after start-up to meet continual bar block feed when spindle operates, this function is strictly limited, and the user can use it when safety is guaranteed.

=0: Spindle clamping and loosing startup in the processing is prohibited

=1: Allowed, but it's prohibited using when safety hasn't be confirmed

B183:=0: Up and down Z axis of water-jet flow special machine tool is on-off

=1: Up and down Z axis of water-jet flow special machine tool is servo/stepping motor

B184:=0: Clamping/loosing pedal switch of spindle is single switch

=1: Clamping/loosing pedal switch of spindle is ganged switch, one switch is clamped and the other one is loosened

B186, B187, B188:=0: When the fine positioning of returning to machine tool reference point starts, the moving axis will move reversely

=1: When the fine positioning of returning to machine tool reference point starts, the moving axis won't move reversely

19#

B191	B192	B193	B194		B196	B197	B198

B191:=1: Check whether spindle has been clamped before start-up=0: Not to check.

B192:=0: The mechanical origin can be only returned after power-on by continually pressing manual feed button, and this axis can move;

=1: Press the manual feed button when returning to mechanical origin after power-on, and this button can be released after start-up of this axis;

B193:=0: After the startup of spindle is checked normal, use external I/O port to input.

=1: After the start-up of spindle is checked normal, use the state recorded in the system.

B194：=0：Time lubrication is on； =1： Timing lubrication is off.

B196, B197, B198:=0: Z axis, Y axis and X axis motor won't use internal contracting brake

=1: The internal contracting brake functions of Z axis, Y axis and X axis are opened

The internal contracting brake will output internal contracting brake release signal after delaying PA4# when the servo enabled is output in one axis or multi-axis system, when servo gives alarm, cancel all internal contracting brake release signals

The internal contracting brake outputs TRF signal of each axis and servo interface

The factory set of 20# is 00000000

21#

				B215	B216	B217	B218

B215：=1：X manual feed key reverse

B216：=1：clear machining time count per loop start

B217：=1：open external key to control movement of each shaft

B218：=1：the piece count and time count are not cleared when the machine is powered on every time =0 clear

The factory set of 21# is 00000000

22#

			B224		B226	B227	B228

B224：=1：cancel continuous transition of block =0：block continuous transition

B226：=1：open G27 piece count function；=0：not opened

B227：=1：not memorize coordinate, cutter compensation can't be input

B228：=1：modify cutter compensation during machining, pause isn't required=0: modify cutter compensation after pause

The factory set of 22# is 00000011

28#

B281		B283		B285	B286		B288

B281：=1：40# input port (XSEL) active high level；=0：active low level

B283：=1：38# input port (*100) active high level；=0：active low level

B285：=1：36# input port (EXEM) active high level；=0：active low level

B286：=1：35# input port (*10) active high level； =0：active low level

B288：=1：33# input port (ZSEL) active high level; =0：active low level

The factory set of 28# is 00000000

29#

B291							

B291：=1：Z-axis returns reference point automatically when alarm is customized (option)

The factory set of 29# is 00000000

Appendix Four Programme illustration

Example: See part as below.

Material : cooper, semi-finished part: forged part, unilateral allowance 1mm,

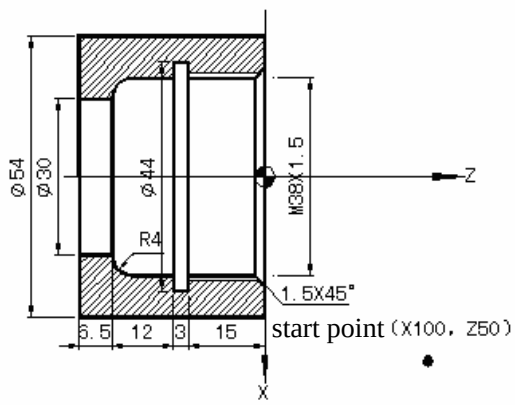
The thread is metric straight thread, pitch 1.5 mm，

1# cutter：internal turning tool(boring tool)

2# cutter：slot cutter(slot width equals to cutter width as 3mm)；

3# cutter：thread chaser

cutter entry point is (X100,Z50)




```
N0010 M03 S1500
N0020 G00 X100 Z50
N0030 T1
N0040 G00 X30
N0050 G00 Z0
N0060 G01 X55 F150                (process end face)
N0070 G01 X39.5
N0080 G01 X36.5 Z-1.5             (chamfer 1.5×45)
N0090 G01 Z-26                    (internal hole turning φ38)
N0100 G03 X30 Z-30 R4 F100        (internal arc turning R4)
N0110 G01 Z-37                    (internal hole turningφ30)
N0120 G00 X28                     (cutter retract in X axes direction)
N0130 G00 Z50                     (cutter retract in Z axes direction)
N0140 T2                          (change internal slot cutter to cut slot)
N0150 G00 X35                     (rapid engage)
N0160 G00 Z-18                    (rapid engage)
N0170 G01 X44 F150                (slotting)
N0180 G00 X36                     (rapid retract)
N0190 G00 Z50                     (rapid retract)
N0200 T3                          (change threading tool to process thread)
N0210 S700
N0220 G00 X36.5 Z2                (rapid engage)
N0230 G86 Z-16 K1.5 I-4 R2 L4      (threading)
N0240 G00 X100 Z80
N0250 M05
N0260 T1
N0270 G00 X100 Z50
N0280 M02
```

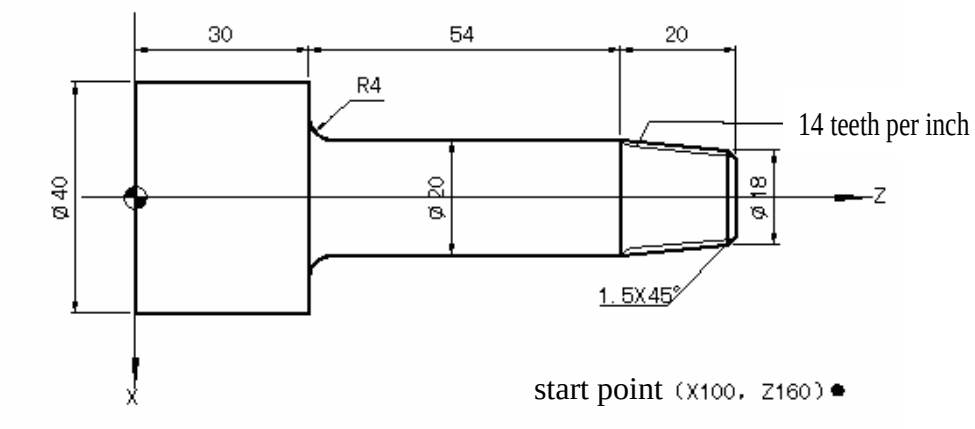
Example two: see part below:

Material :45#, semi-finished part: forged part, unilateral allowance: 1mm,

The thread is inch size tapered thread， pitch is 1/14”

1# cutter：external turning cutter, 2# cutter：external thread chaser

cutter entry point (X100,Z160)




```
N0010 M03 S1000
N0020 M08
N0030 G00 X100 Z160
N0040 T1
N0050 G00 X44 Z30.2                 (rapid engage)
N0060 G01 X30 F120                  (rough turning end face)
N0070 G00 Z107                      (rapid retract)
N0080 G00 X18.4                     (rapid engage)
N0090 G01 Z104 F120                 (slow engage)
N0100 G01 X20.4 Z84                 (rough external taper turning, dial. allowance 0.4)
N0110 G01 Z34                       (rough external turning φ20)
N0120 G02 X28 Z30.2 R3.8 F80        (rough turning R4)
N0130 G01 X40.2 F120                (rough end face turning)
N0140 G01 Z10                       (rough external turningφ40)
N0150 G00 Z104                      (rapid retract)
N0160 S1500
N0170 G00 X24                       (rapid engage)
N0180 G01 X10 F120                  (fine turning end face)
N0190 G01 X15
N0200 G01 X18 Z102.5                (chamfer1.5×45)
N0210 G01 X20 Z84                   (fine taper turning )
N0220 G01 Z34                       (fine external turning φ20)
N0230 G02 X28 Z30 R4 F80            (fine turningR4)
N0240 G01 X40 F120                  (fine turning end face)
N0250 G01 Z10                       (fine external turningφ40)
N0260 G00 X100 Z160                 (rapid retract)
N0270 T2                            (change thread chaser to threading)
N0280 S700
N0290 G00 X17.8 Z106                (rapid engage)
N0300 G87 X20 Z84 K14 I4 R1.96 L8   (turning taper thread)
N0310 G00 X100 Z160
N0320 M05
N0330 M09
N0340 T1
```

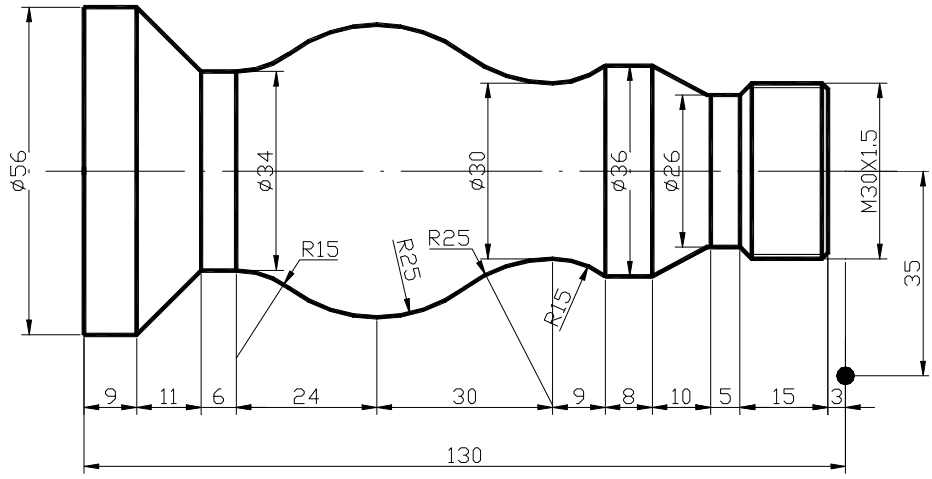
N0350 G00 X100 Z160
N0360 M02

Example: 3. the part in the below, total three cutters

1# cutter：90°external turning cutter，2# cutter：5mm slotting cutter（5mm），

3# cutter ：60° threading cutter

semi-finish part φ60×150， material： aluminum



programme：

N0010 G00 X70 Z130	
N0020 M03 S800	
N0030 G01 Z127 F80	(slow engage)
N0040 X-0.5	(turning end face)
N0050 G00 Z130	(rapid retract)
N0060 X56.2	(rapid retract)
N0070 G01 Z0 F80	(rough external turningφ56)
N0080 G0 X58	(rapid retract)
N0090 Z130	(rapid retract)
N0100 G01 X50.5 F80	(slow engage)
N0110 Z14	(rough external turning)
N0120 G00 X52	(rapid retract)
N0130 Z130	(rapid retract)
N0140 G01 X44 F80	(slow engage)
N0150 Z70	(rough external turning)
N0160 G00 X46	(rapid retract)
N0170 Z130	(rapid retract)
N0180 G01 X40 F80	(slow engage)
N0190 Z70	(rough external turning)
N0200 G00 X42	(rapid retract)
N0210 Z130	(rapid retract)
N0220 G01 X36.2 F80	(slow engage)
N0230 Z75	(rough external turning)
N0240 G00 X38	(rapid retract)
N0250 Z130	(rapid retract)
N0260 G01 X28.5 F80	(slow engage)
N0270 X30.5 Z125	(chamfer)
N0280 Z104	(rough external turning)
N0290 G00 X90	(rapid retract)
N0300 Z200	(rapid retract)
N0310 T02	

```
N0320 M03   S400
N0330 G00 Z107                      (rapid engage)

N0340 X32                           (rapid engage)
N0350 G01 X26.2 F20
N0360 G00 X52                       (rapid retract)
N0370 Z20                           (rapid retract)
N0380 G01 X34.2 F20                 (rough external turning)
N0390 G01 X52 F80                   (slow retract)
N0400 G00 Z200                      (rapid retract)
N0410 T3 S1200
N0420 G00 X32 Z127
N0430 G01 X30 F80                   (slow engage)
N0440 Z114                          (fine external thread turning φ30)
N0450 X26 Z112                      (chamfer)
N0460 Z107                          (fine external turningφ26)
N0470 X36 Z97                       (fine taper turning)
N0480 Z89                           (fine external turningφ36)
N0490 X54 Z50                       (rough external turning)
N0500 X38 Z26                       (rough external turning)
N0510 Z20                           (rough external turning)
N0520 X58 Z9
N0530 G00 Z97                       (rapid retract)
N0540 G01 X36 F80                   (slow engage)
N0550 Z89                           (slow engage)
N0560 G02 X30 Z80 R15                (fine turning clockwise R15)
N0570 G02 X40 Z65 R25                (fine turning clockwise R25)
N0580 G03 X40 Z35 R25                (fine turning counter clockwise R25)
N0590 G02 X34 Z26 R15                (fine turning clockwise R15)
N0600 G01 Z20                       (fine external turningφ34)
N0610 X56 Z9                        (fine taper turning)
N0620 Z0                            (fine external turning φ56)
N0630 S400
N0640 G00 X58                       (rapid retract)
N0650 Z132                          (rapid retract)
N0660 X30                           (rapid engage)
N0670 G86 Z110 K1.5 I4 R1.35 L8     (threading）
N0680 G0 X70                         (rapid retract)
N0690 Z200                          (rapid retract)
N0700 T1
N0720 M02
```

Nanjing Washing CNC technology CO.,LTD

Address: Centralized Industrial Zone, DongShan Bridge, JiangNing District, NanJing City, JiangSu Province

Telephone: (025)87170996 87170997 87170998

Fax: (025)52627632

Post Code: 211153

Email:njwxcnc@163.com