In this manual, we will try to describe various matters relating to the operation of the system. The limitation of space and the specific use of the product are impossible to describe in detail in the system, which is not necessary to do and/or cannot be done. Therefore, none of the items specified in this manual are deemed to be "impossible" or "not permitted".

The copyright of this manual shall be owned by GXK CNC SYSTEM company, and any unit or individual shall be entitled to the legal liability of the company.

Preface

Dear clients:

We are deeply honored and grateful for the products you have chosen.

This manual details the programming, operation and installation of 1000TC lathe CNC SYSTEM.

In order to ensure product safety, correct and efficient operation, please read the manual carefully before installing and using the product.

Safety Warning

Improper operation will cause an accident. A person must be qualified to operate the system。

Special note: system power installed on the chassis (inside) is a dedicated power supply only for the CNC system made by our company. Users are prohibited from using this power for other purposes. Otherwise, there will be great danger!

Considerations

■ Transportation and storage

1. No more than six floors of product packaging;
2. Do not climb, stand or place heavy objects on product package.
3. Do not use cable dragging or handling products connected with the product;
4. Collision, scratch panel and display screen are strictly prohibited;
5. Product packaging should avoid damp, sun and rain.

■ Check the package

1. Please confirm if it is the product you purchased after opening the package;
2. Check whether the product is damaged during transit;
3. Check the list to make sure the parts are complete and without damage;
4. Please contact us if there is any discrepancy in product model, lack of attachment or transport damage.

■ Wiring

1. The personnel involved in wiring and inspection must be professionals with corresponding abilities;
2. The product must be reliable grounding, grounding resistance should be less than 0.1 Ω, cannot use neutral line (zero line) instead of the ground;
3. The wiring must be correct and firm to avoid product failure or unintended consequences;
4. The surge absorbing diode connected with the product must be connected in accordance with the specified direction, otherwise the product will be damaged.
5. The product power supply must be cut off before plugging the plug or opening the product cabinet.

■Maintenance

1. Cut off the power supply before repair or replacement of components;
2. The fault should be checked during short circuit or overload, and the failure can be restarted after troubleshooting.
3. Do not power the product frequently, and if the power is off, it must be renewed for at least 1min.

The Statement!

This manual try to explain all kinds of different contents, but, due to the possible nature involved
Too much to explain all the actions that can or cannot be done. Therefore, this manual does not
The content of a special note may be considered unusable.

Warning

Prior to installation and connection, programming and operation of this product, the product manual must be read in detail,And the instruction manual of the machine tool manufacturer, strictly according to the requirements of the manual and instruction,Otherwise, it may result in product, machine tool damage, scrap and even personal injury.

Pay Attention

Product features described in this manual, technical indicators (such as accuracy, speed, etc.) only for this product, the installation of the the products of nc machine tools, the function of the actual configuration and technical performance is determined by the design of the machine tool factory, CNC machine function configuration and technical indicators will be subject to the use of the machine tool manufacturers specifications;
Although the system is equipped with standard machine tool operation panel, the function of each key of standard machine tool panel is defined by PLC program (ladder diagram). The function of the key in the machine tool panel is described in the standard PLC program. Please pay attention!

* The contents of this manual are subject to change without prior notice.

Security Responsibility

The safety responsibility of the manufacturer

- the manufacturer shall be responsible for the design and structure of the control and/or control of the supplied numerical control system and the annex to the accompanying supply.

- the manufacturer shall be responsible for the security of the supplied numerical control system and the accompanying supplies.

The manufacturer shall be responsible for the use of information and advice provided to the user.

User's security responsibilities

-- the user should learn and train the security operation of the CNC system and know and master the contents of the safe operation.

-- the user shall be responsible for the increase, transformation or modification of the original numerical control system, the security of the attachment, and the resulting danger.

- the user shall be responsible for handling, adjusting, maintaining, installing and transporting products without the prescribed manual.

* this manual is for the end user. We sincerely thank you for your friendly support to our company when using our products.

Directory

Chapter I Product introduction

1.1 Product introduction

The 1000TC system can control 4 feed shafts (including the C axis), 1ms high-speed interpolation, and 0.1 mu m control precision to improve the efficiency, precision and surface quality of the part manufacturing.

X, Z, Y, A four-axis control, axis of name and type can be defined by customer

*** 1ms interpolation cycle, control accuracy 1 mu m, 0.1 mu m optional**

*** maximum speed of 60m/min (maximum speed of 24m/min for 0.1 mu m)**

*** adaptive servo spindle can realize asynchronous servo Machine-tool spindle milling automatic switching and electromagnetic switching machining.**

*** built-in multi-plc program can realize automatic upper and lower feeding function.**

*** it also supports the function of carpenters and milling and carving, and the system software includes two perfect cutting techniques**

*** support double - knife and one - knife processing molding, support double - knife and one - knife split repeatedly turning**

*** support statement macro code programming to support macro program calls with parameters**

*** breakpoint memory function, the process interrupt the electrical location will not be lost**

*** graphic display, real-time machining trajectory tracking display.**

*** the import of processing files supports manual editing of G instructions, U disk G code files and DXF files which are directly read , imported into Chinese DBF files.**

*** simultaneously control 4 step/servo motor, support 4 axis G code instruction**

*** external handheld box function**

*** support for metric/English programming, with automatic knife and tool life management**

*** support Chinese and English display, selected by parameters**

*** with USB interface, support for USB file operation, system configuration and software upgrade**

*** key file backup and restore, system parameter backup and restore function**

*** supports the function of the four-station electric tool holder**

*** 1 road 0V ~ 10V analog voltage output**

*** 1 hand wheel input, supporting handheld unit**

*** 16 points universal input /16 point general**

1.2 Technical specifications

Number of control shaft

* control axis number: 4 axis (X, Z, Y, A)

* linkage axis: 4 axis feed shaft function

* minimum input increment: 0.001mm

* minimum instruction increment: 0.001mm

* maximum stroke: plus or minus 99999999 x minimum instruction increment

* rapid movement speed: maximum 60m/min

* rapid rate: F0, 25%, 50%, 100% of cet 4 real-time correction

* feed ratio: 0 ~ 150% of total 16 real-time correction

* interpolation: linear interpolation, circular interpolation (supporting three-point arc interpolation)

* automatic chamfering function

Acceleration function

* cutting feed : the front plus deceleration is linear, the front plus decelerate S type

* rapid movement: the front plus deceleration is linear, the front plus decelerate S type

* The starting speed, stopping speed and deceleration time of the acceleration deceleration are set by parameters

machine tool spindle function

* 1 road 0V ~ 10V analog voltage output

* Machine tool spindle speed: can be given by S code, speed range 0r/min ~ 9999r/min

* Machine tool spindle ratio: 50% ~ 120% of the total 8 level real-time correction

Function of cutting tool

* tool life management

* the knife calibration: the fixed point calibration, the test cutting calibration, back reference point calibration

* knife partial execution mode: modification of coordinate mode, tool movement mode

Precision compensation

* reverse clearance compensation

* The memory type pitch error compensation

The PLC function

* two-stage PLC program, up to 4,700 steps, and the 1st level program refresh cycle 8ms

* PLC program communication download

* support PLC warning and PLC alarm

* supports multiple PLC programs (up to 20), and the currently running PLC programs can be selected

* basic I/O: 16 input /16 output

The man-machine interface

* 7.0-inch widescreen LCD with a resolution of 800 x 480

Chinese, English and other languages

* 2d tool track display
* real-time clock

Operation management

* operation mode: edit, auto, input, machine tool back zero, hand wheel/single step, manual, program return to zero
* multi-level operation permission management
* alarm log

The program to edit

* program capacity: 56MB, 400 procedures (including subroutine, macro program)
* edit function: program/program segment/word retrieval, modification, delete, copy, paste
* program format: ISO code, support statement macro code programming, support relative coordinates, absolute coordinates, and mixed-coordinate programming

Communications functions

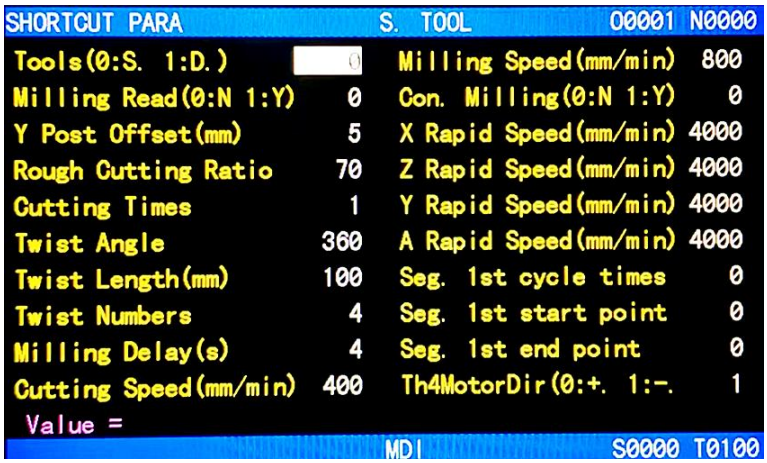
* USB: U disk file operation, U disk file direct processing, support PLC program, system software U disk upgrade

Safety features

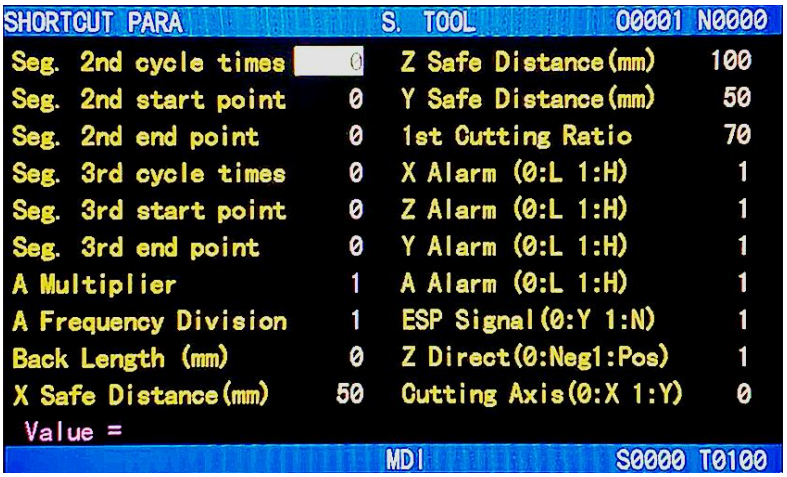
* emergency stop
* hardware travel limit
* software travel inspection
* data backup and recovery

Chapter 2 General parameters

2.1 Quick parameters



Lathe knife (0: single-blade 1: double knife)	0	milling into the speed (mm/min)	800
Milling cutter reading mode (0: no 1: is)	0	continuous milling (1:0: closing)	0
After Y the offset (mm)	5	X axis fast speed (mm/min)	4000
The ratio of the coarse knife	70	Z axis fast speed (mm/min)	4000
The Lathe cut times	1	Y axis fast speed (mm/min)	4000
The angle of twist	360	A axis fast speed (mm/min)	4000
The length of twist (mm)	100	Seg. 1st cycle times	0
The twist number	4	Seg. 1st start point	0
Milling machine tool spindle delay (S)	4	Seg. 1st end point	0
Lathe cut feed speed	400	Th4Motor DIR (0: +. 1: -.)	1



Seg. 2nd cycle times	0	Z axis security margin (mm)	100
Seg. 2nd start point	0	Y axis security margin (mm)	50
Seg. 2nd end point	0	1st Cutting Ratio	70
Seg. 3rd cycle times	0	X axis alarm electrical level	1
Seg. 3rd start point	0	Z axis alarm electrical level	1
Seg. 3rd end point	0	Y axis alarm electrical level	1
A Multiplier	1	A axis alarm electrical level	1
A Frequency Division	1	ESP Signal (0: Y 1: N)	1
Backknife length (mm)	0	Z Direct (0:Neg 1:Pos)	1
X axis security residual (mm)	50	Cuttting Axis (0:X 1:Y ）	0

FT Cut Enable(0:N 1:Y)	0	Cut oneside (0:NO 1:YES)	0
FT Cut Remain	0	MACH_Cancel (0:NO 1:YES)	0
FastCutEnable	0	Don't use backup parameter	0
Mill Numbers	0	Don't use backup parameter	0
Mill Delay	0	Don't use backup parameter	0
Mill Axis (0:X 1:Y)	0	Don't use backup parameter	0
Cross Mill (0:NO 1:YES)	0	Don't use backup parameter	0
CNC Code M70(0:NO 1:YES)	0	Don't use backup parameter	0
Mill by Head (0:NO 1:YES)	0	Don't use backup parameter	0
Cut long Axis (0:Z 1:Y)	0	Don't use backup parameter	0

Chapter 3 basic operations

3.1 Coordinate

When the 1000TC lathe system is started, the screen will show X0.000 and Z0.000, and the X and Z here are the coordinates we used to say, or X and Z axes. Any friend who has operated a normal lathe should know the concept of "big drag plate" and "drag-board", and the X-axis is a large drag board in a normal lathe, and the z-axis is a big drag plate. In the numerical control machine, the direction of the direction of the workpiece is the direction of the direction of the workpiece.

3.2 Move X and Z axis manually

1. Press the manual key 

, and the "manual mode" will be displayed at the bottom of the screen, indicating that the current system works in "manual mode".

2. Press the z-axis negative key 

. At this point, the Z coordinate in the screen changes, and the big drag of the machine moves in the direction of the machine tool spindle (chuck). The beginner should be careful to move too fast and hit the chuck.

3. Press the Z axis forward 
, and the Z coordinate in the screen changes incrementally, and the big drag of the machine moves towards the end. The beginner should be careful to move too fast and too fast into the tail seat.

4. The function of the key 

is to move the axis of the manual movement fast/slow. When press the 

key, and the indicator light in the upper left corner is lit, the manual movement of the axis is to move quickly, then should be careful not to cause a collision. When

pressed 

again, the indicator in the upper left corner of the button goes out, indicating that the axis of the manual movement is moving at a slower speed, which is safer.

5. Manual movement of the X axis is the same as the Z axis.

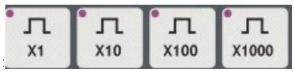
3.3 The handwheel mode moves X and Z axis

1. press the handwheel way 

, the bottom of the screen will show "the handwheel way" at the same time, the current system works as "the handwheel way",then can use the handwheel which installed on the panel to move coordinates. Use handwheel can enable coordinate movement free control, so this often used in the knife calibration operation.

2. Press the Z axis moving key 

with the shape of the handwheel  , then rotate the handwheel slowly and counterclockwise. The z-axis coordinates in the screen are decreasing, and the big towing plate moves in the direction of the machine tool spindle (chuck). If the handwheel is shaken clockwise, the Z axis coordinates are changing, and the big slabs move in the direction of the stern.

3. The selection of the multiplier key 

can enable the handwheel to shake the button to move the coordinates as it is multiple number. X 1 is 0.001 x 1=0.001 mm, which is the handwheel shaking a lattice, the coordinate moves 0.001mm, the x 100 is 0.001 x 100= 0.1mm, which is the handwheel shaking one lattice, the coordinate moves 0.1mm, the other same. When you press 

, if there is no response, it is likely that the machine tool manufacturers block the ratio of this file to prevent the stepping into the motor. Don't misunderstand that the system is wrong.

4. The rotation of the X axis hand wheel is similar to the Z axis.

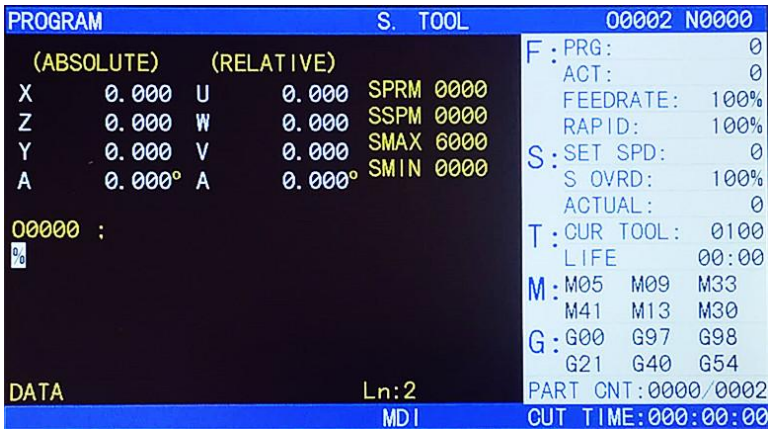
3.4 The input mode opens the machine tool spindle

1. Press the input mode key 

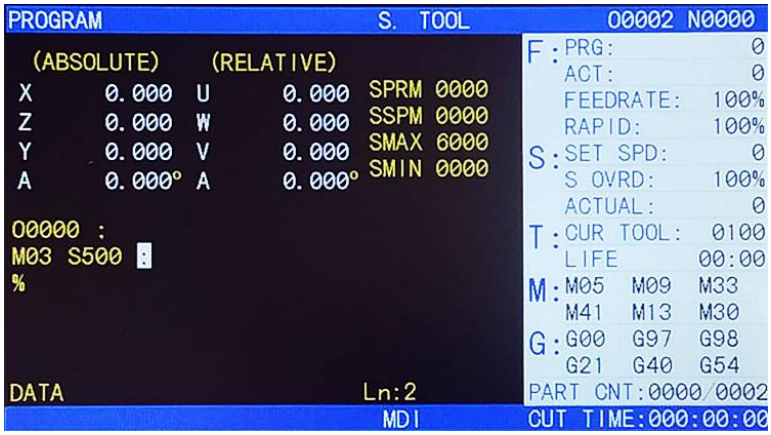
, and the "input mode" will be displayed at the bottom of the screen, indicating that the current working mode of the system is "input mode", which means "MDI".

2. Press 

again and again until the screen shows "program status" page, as shown below:



3、If you operate the machine tool spindle function for stepless speed regulation, press M[、0、3、S]、5、0、0、输入 IN, as shown below:



at last press 循环启动, At this point, the machine tool spindle is rotation at a speed of 500 revolutions per minute.

4、If the machine tool you operate, the machine tool spindle of the ordinary is a multi-speed motor, press M[、0、3、S]、0、1、输入 IN、循环启动, At this point, the machine tool spindle is rotation at the first speed of the main motor. If your main motor is two-speed, you can also press S]、0、2、输入 IN、循环启动 Rotate the machine tool spindle to the second speed。

5、If the machine tool you operate the machine tool spindle of the ordinary single-speed motor, press M[、0、3、输入 IN、循环启动, The machine tool spindle is spinning forward.。

6、Press M[、0、5、输入 IN、循环启动, The machine tool spindle is stop。

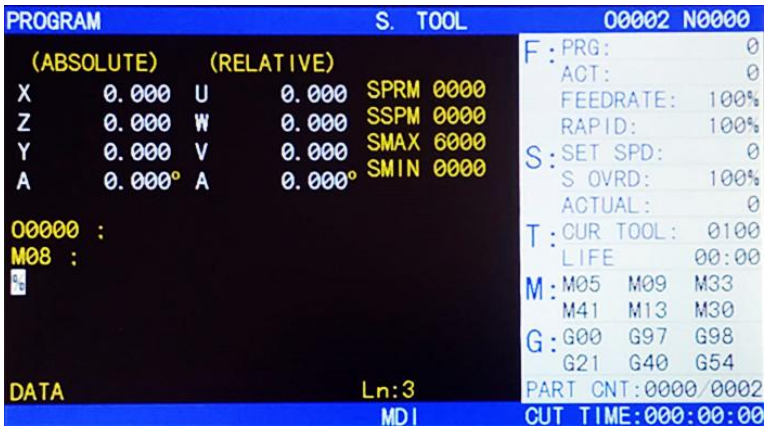
7、Switch to "manual mode", press 主轴正转, The machine tool spindle is moving forward, Press 主轴停止, The machine tool spindle stop, press 主轴反转, The machine tool spindle runs in reverse 。

When there is the first time to turn on the power for the machine， Press 主轴正转 and 主轴反转 The machine tool spindle will not running ,Because you haven't specified a running speed for the system. In the input mode, if a speed is running, such as the S500, it will be valid until the machine is switched off.

3.5 The input mode opens the Milling cutter spindle

1 、Press the input mode key MDI , The "entry mode" is displayed at the bottom of the screen, indicating that the current system works in "input mode", which means "MDI".

2 、Press 程序 PRG key over and over again， Until the screen shows "program status" page, as shown below:



3 、Press M [、 0 、 8 、 输入 IN 、 循环启动 , Mill spindle working. Press M [、 0 、 9 、 输入 IN 、 循环启动, Mill spindle stop working.

3.6 Mechanical coordinate return to zero

1 、Press 机床零点 key,Switch the working mode of the system to the "mechanical zero" mode;

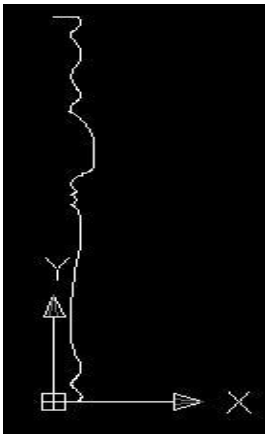
2 、 Determine that the X and Z axis will not have the rear seat, tools, sundries, and iron debris And so on, then press X 、 Z ;

3 、X, Z move fast to the biggest position of machine tool, until hitting the back to zero switch,which installed in the axis position of the biggest， X、Z will continue to move forward with low speed, until out of the back to zero switch to reach back to zero position, and the back to zero indicator light on the panel system on, at the same time set the X, Z axis coordinate and machine coordinate absolutely zero at the same time, the machine back to zero finish.

3.7 CAD file direct import

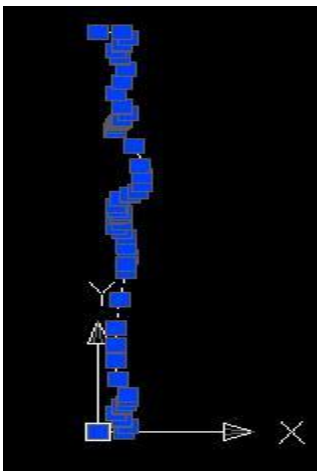
1. drawing

Draw the shape diagram according to the actual size and use the "translation" instruction  , Shift the starting point to the zero point position of the CAD,
As shown in figure:



2. save as CAD R12/LT2 DXF
If there is a spline in the graph, you must save this file as CAD R12/LT2 DXF file。

3. Reopen the savedCAD R12/LT2 DXF file, Select the change to see the spline curve.



4. Using decomposition instructions  , Decomposes the drawing。
5. Convert the graph to multiple lines, and operate as follows:

Enter“PE” command,then press enter

```
命令: *取消*
命令: pe
PEDIT 选择多段线或 [多条(M)]: m
```

Enter “M”to select if more than 1 line, press enter，and then select all the lines

```
选择对象: 指定对角点: 找到 784 个
选择对象:
是否将直线和圆弧转换为多段线? [是(Y)/否(N)]? <Y>
```

Press enter

```
选择对象:
是否将直线和圆弧转换为多段线? [是(Y)/否(N)]? <Y>
输入选项 [闭合(C)/打开(O)/合并(J)/宽度(W)/拟合(F)/样条曲线(S)/非曲线化(D)/线型生成(L)/放弃(U)]:
```

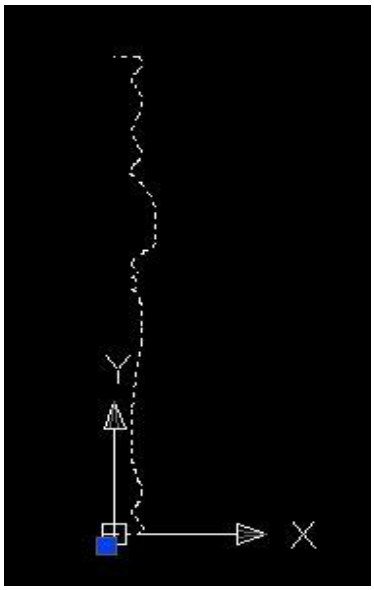
Enter J then coalescing，press enter

```
弃(U)]: j
合并类型 = 延伸
输入模糊距离或 [合并类型(J)] <0.0000>: 0.1
```

Input fuzzy distance 0.1 or 0，press enter

```
输入模糊距离或 [合并类型(J)] <0.0000>: 0.1
多段线已增加 783 条线段
输入选项 [闭合(C)/打开(O)/合并(J)/宽度(W)/拟合(F)/样条曲线(S)/非曲线化(D)/线型生成(L)/
```

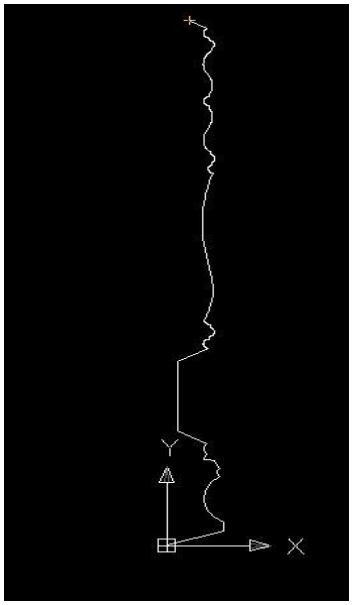
At this point, multiple lines have been converted.



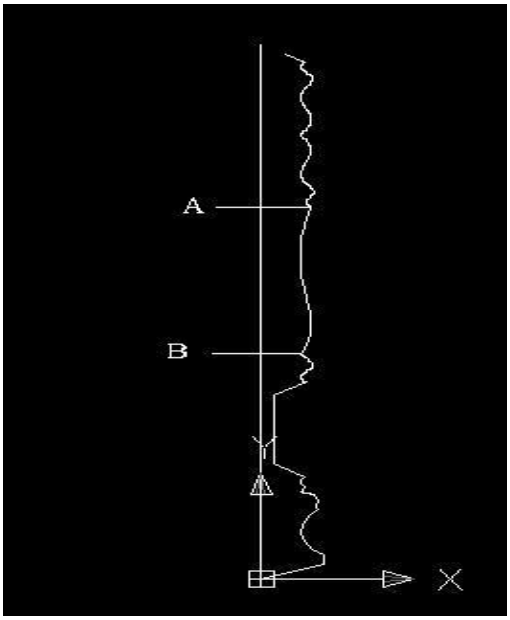
6. Finally, the transformed multi - segment line is saved as CAD R12/LT2 DXF file， Copy to GXK1000TC controller with U disk and process.

3.8 Milling function

The system supports simple hemp milling function.



As shown in figure DXF graphic car processing, suppose we need to process the hemp with a section between AB two in the image below, and after opening the file, use the decomposition **instruction**  ,Smash the graphics.



Draw the vertical line of the position to the end of the line as the auxiliary line, and then the vertical line of the auxiliary line at the point AB two, then trim.

The line in the position of processing starting point to finish point includes: starting line, machining shape of processing section and back cutting line, all other figures are deleted. The rest of the graph is the shape of the figure milling AB.

Enter PE instructions according to the merge method introduced in the previous section, then the remaining lines will be merged into a multi-section line by the prompt, and exit after saving into R12 format.

Enter parameters, adjust. Modify the ratio parameters of the hemp and the parameters of the flax.

Twist proportion:

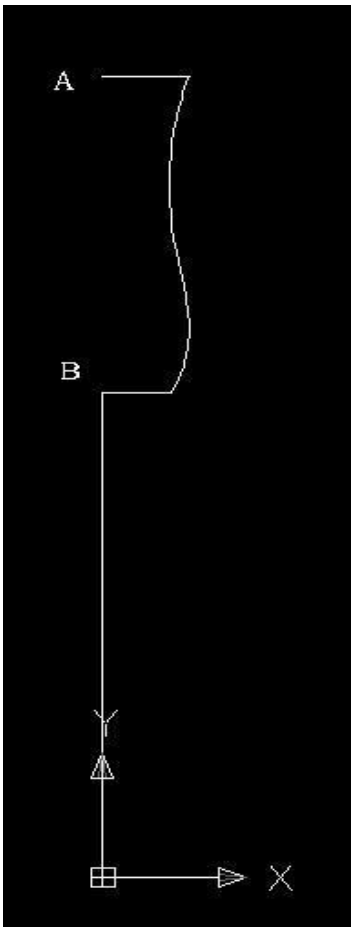
The relationship between the running length of milling time axis and the rotation Angle

Twist number:

The relationship between the Angle of the hemp and the number of processing bars is required.

After setting the parameters, enter the processing interface, switch to the woodworking milling mode, read the graphics, the knife, the processing.

The processing pattern of the pruning flowers is shown as follows:



Chapter 4 1000TC system calibration

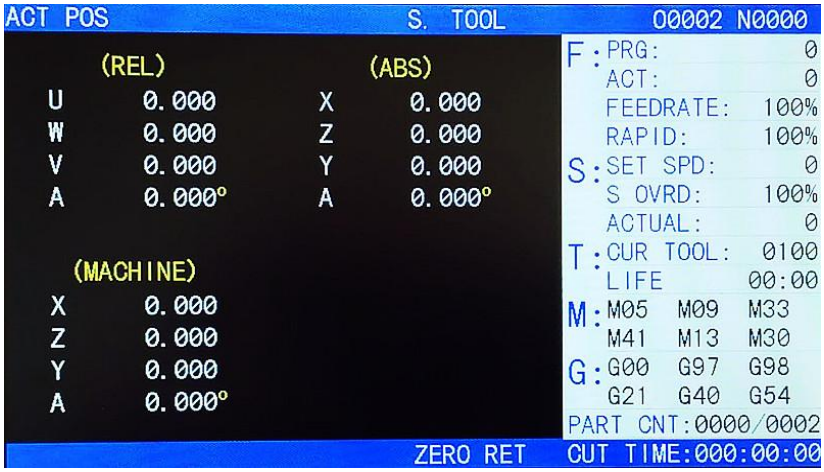
4.1 What is knife repair and knife calibration

Knife-edge, also known as knife offset, is the position of the system tool relative system coordinates by means of the knife calibration operation. Tool installed on the head when the operator put the prepared, They could not set every knife installation location (coordinates) as the same, even if each tool can be installed in the same position, you also should tell the numerical control system the tool relative to X, Y, Z coordinate exactly on the coordinate values. Therefore, to determine each tool in the X, Y, and Z coordinates the coordinates, is the sword, and 1000 tc in the operating system to determine the cutting tool on the X, Y, Z coordinate position, the process of call for the knife calibration.

4.2 knife calibration

knife calibration and setting safety point

knife calibration ： Let the standard workpieces which need to be cutted fixed, and then manually, in turn, aim the X/Y point on the edge of the biggest contour workpiece outside, the Z axis moved to processing starting location, and then hold the knife key and press enter key twice at the same time, the system will set the current edge location (i.e., processing starting location) to 0, then the system back to safety positon.



Compensation: the system back to the knife start point, moving the X/Y/Z axis to the correct position according to the actual measured value, and restarting the knife calibration. For example, the actual measurement of the diameter the outer circle is 1mm, and we do the following operation according to the handwheel key, the system steps in and selects each step 0.1 mm, moves the work direction five times, and redo the knife calibration operation.

Set the safety point: after the knife calibration finished, set a safe parking position after processing, so that after each processing, the system will return to this point and stop the movement.

MILL calibration： Move the XY axis and align the rotation center of the milling cutter at the zero point position of the tool.

(continuous milling and closing) move the XY axis, align the rotation center of the milling cutter to the zero position of the tool, press the knife, and then determine.

When the above setup steps are over, press [start] to process the keys directly.

On the switch setting page, the opening and closing status of the parameters, procedures, automatic sequence Numbers can be displayed, and the page displays as shown below:

4.3 Permissions Settings and modifications

In order to prevent, CNC machining program is malicious modification parameters, 1000 tc deliver the function of permissions, password is divided into 4 grade, from high to low were grade 2 (machine tool factory), 3 (equipment management level), grade 4 (engineer), 5 (processing), CNC operation level of the current permissions Settings page of "current operation level:" for display.

Level 2: machine tool manufacturer, allow to modify the state parameters of CNC, data parameters, screw patch data, knife filling data, edit part program, transmission PLC ladder diagram, etc.

Level 3: the initial password is 12345, which allows the modification of the state parameters of CNC, data parameters, knife filling data, and compilation

Part procedures.

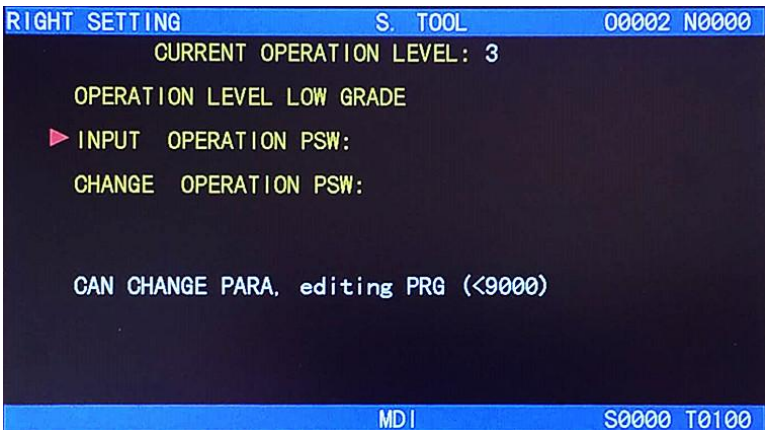
Level 4: the initial password is 1234, can modify the knife's complement data (carry on the knife operation), macro variables, edit the part program, cannot modify the state parameters of CNC, data parameters and the conch data.

Level 5: password-less level for the operation of the machine tool operation panel, immutable knife to fill the data, choose the parts program, an editor, do not modify the state of the CNC parameters, data, and screw data.

4.3.1 Enter the password operation method

Press： 

，then press  or  key, Find the "permission Settings" page, as shown below:



Press MDI key， Enter the input mode, press ⇧ or ⇩ Move the cursor to the "enter password" position, then enter your password, such as: 1234, finally click 输入 IN ， If the password is correct, the password is passed and the password is authenticated.

4.4Preparation of processing procedures

Press 编辑 k ey ， Enter edit mode, press 程序 PRG ， then press ▤ or ▤ ey to find program page as shown below:

Press O␣ 、0、0、0、1， 换行 EOB, then The new program, called 0001, is more editable.

Chapter 5 G code

5.1 Definition of coordinates

1000TC using a rectangular coordinate system of the X axis and Z axis, the X axis vertical to the shaft axis and Z axis parallel to the spindle axis direction, close to the direction of the workpiece is negative direction, away from the direction of the workpiece as the positive direction.

5.2 G code

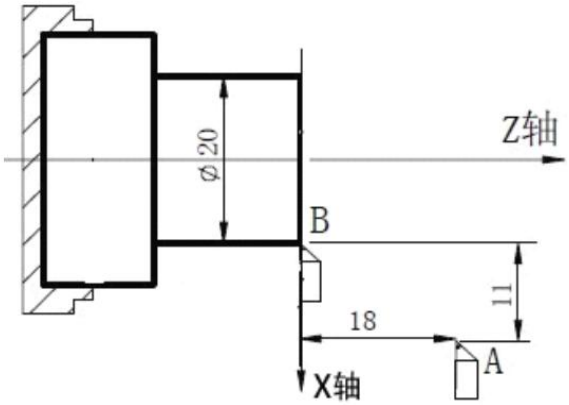
5.2.1 Rapid positioning： G00

Code format： G00 X/U Z/W ；

1: G00 x 50 Z2; // this code means: 1000TC is the maximum speed limit set by the machine factory home, moving to X coordinates: 50.000, and Z coordinate is 2.000.

2: G00 W U - 10-20 / / this code means: 1000 tc to vendors would set the fastest speed limit, the X axis from the current coordinates on fast moving in the direction of the X axis negative 10.000 mm, Z axis from the current coordinates on fast moving in the direction of Z axis of negative 20.000 mm. Assuming the current coordinates are: X100.000,Z100.000, then the coordinate stops at: X90.000, Z80.000, after g00u-10w-20.

Example: the cutting tool moves quickly from point A to point B. The diagram below:



The cutting tool moves quickly from point A to point B, and the following code is correct:

G0 x 20 Z 0; (absolute coordinate programming) or

G0 U - 22 W - 18; (relative to coordinate programming) or

G0 X20 W - 18; (mixed-coordinate programming) or

G0 U minus 22 Z 0; (mixed-coordinate programming)

Why is it that the X distance from point A to point B is 11.0mm, while the code is written as: G0, u-22, w-18? Because lathe is X to the diameter of the programming, that is to say, the unilateral cutting 11 mm, due to the rotation of the work but in cutting the 22 mm diameter, and U - 22 in order, the actual X axis to negative feed only 11 mm.

5.2.2 Linear interpolation G01

Code function: trajectory for a straight line from start to finish, in our actual processing G01 role is cutting cylindrical, flat end face, cant go, straight Angle and cutting hole everything straight line cutting. The code for G01 is the same as G00, but its cutting speed is specified by the back F.

Such as: G98;

G01 Z - 22 F100. // the z-axis is given at 100mm per minute to feed to z-220,000 coordinates. (G98 per cent to)

G99;

G01 Z - 22 F0.1; The //Z axis is transferred to the z-220,000 coordinate with each rotation of the spindle to the speed of 0.1 mm. (G98 per cent to)

5.2.3 Every minute G98 Every turn of feed G99

Code format: G98 G98 G01 X/U Z/W F__;

Code power generation: the G98 is the modal G code with mm/min (how many millimeters per minute). Modal means: when you specify G98 once, it defaults to G98 in the code area before the G99.

Code format: G99 G01 X/U Z/W F__;

Code function: the G99 is the modal G code with the change of the mm/spindle per unit to the given cutting speed of the unit. Modal means: when you specify G98 once, it defaults to G99 in the code area before the G98. The premise of using the G99 instruction is that the spindle of the machine must be installed with a threaded encoder or the servo spindle is installed, that is, the speed of the spindle motor must be read by the system.

G99 ；

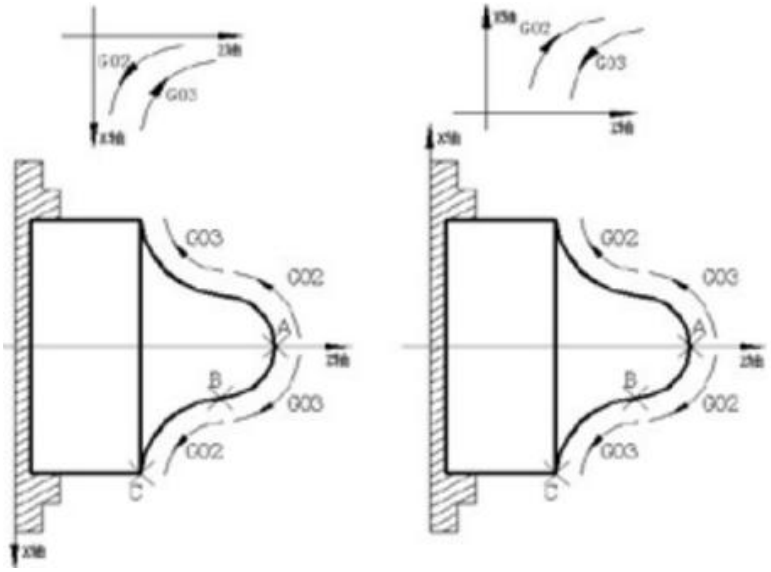
M03 S1500 ；

T0101 ；
G00 X50 Z2
G01 Z-20 F0.07; …。

5.2.4 The circular arc interpolation G02 、 G03

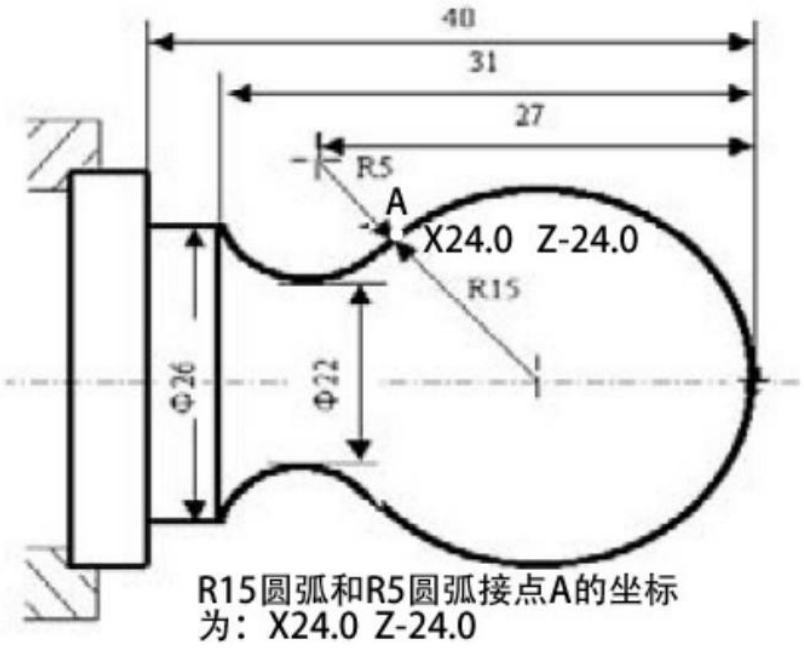
Code format: G02 (G03) X/U Z/W R _ F_

Code function: G02 code trajectory for clockwise from start to finish (rest after coordinate system)/counterclockwise (former head coordinate system), the current economical lathe bed CNC lathes, mostly former head coordinate system. The arc is as shown in figure 3-5. The trajectory of the G03 code is counterclockwise from the starting point to the end (rear point coordinate system)/clockwise (the front of the blade coordinate system). The trajectory is shown in the figure below.



Tool frame (slant bed machine) **after frame (flat bed machine)**

Programming example:



Application: O0001

G99; (defined for each transfer)

G0 X40 Z5. (quick positioning, close to work position)

M03 S 8 00; (main shaft opens, 800 RPM per minute)

G01 X0 Z0 F 0.1; (near the origin of the workpiece, go to the first section of the arc starting point)

G03 U24 W-24 R15 f05; (cutting R15 circular section)

G02 X26 z-31 R5 f05; (cutting R5 circular section)

G01 Z - 40; (cutting phi 26 outer circle)

X40 Z5. (return to starting point)

M30 (program end);

5.2. 5 The pause M00

When the program runs to M00, stop and stop running the next program, pause, and then try again press 

,The program continues to run down.

Example:

Application: O0001

G99; (defined for each transfer)

G0 X40 Z5. (quick positioning, close to work position)

M03 S 8 00; (main shaft opens, 800 RPM per minute)

G01 X0 Z0 F 0.1; (near the origin of the workpiece, go to the first section of the arc starting point)

M00; (the program stops at this point, waiting for the loop to start up and then run the program again)

G03 U24 W-24 R15 f05; (cutting R15 circular section)

G02 X26 z-31 R5 f05; (cutting R5 circular section)

G01 Z - 40; (cutting phi 26 outer circle)

X40 Z5. (return to starting point)

M30 (program end);

5.2.6 delay G04

Code format: G04 P; or

G04 X; or

G04 U; or

G04;

Code function: the axis movement stops, does not change current G code mode and maintains data, state, after a given time, then executes the next program segment.

Code description: G04 is non-modal G code;

The G04 delay time is specified by the code word P, X, or U;

The range of P value is 0 ~ 99999 (unit: ms).

The code range of X and U is 0 ~ 9999.999 X minimum input increment (unit: s).

Notes:

When P, X and U are not entered, the program segment is stopped exactly.

The "P", "X" and "U" cannot appear in the same program segment.

Example:

Application: O0001

G99; (defined for each transfer)

G0 X40 Z5. (quick positioning, close to work position)

M03 S 8 00; (main shaft opens, 800 RPM per minute)

G01 X0 Z0 F 0.1; (near the origin of the workpiece, go to the first section of the arc starting point)

G04 X5.0; (the program runs to this time with a delay of 4.0 seconds and then continues to run the next program)

G03 U24 W-24 R15 f05; (cutting R15 circular section)

G02 X26 z-31 R5 f05; (cutting R5 circular section)

G01 Z - 40; (cutting phi 26 outer circle)

X40 Z5. (return to starting point)

M30 (program end);

5.2.7 G28 The first reference point instruction of the machine tool

Code format: G28 X/U/W __ __ Z;

Code function: start from the starting point, and then return to the machine tool zero at the intermediate points specified by X/U and Z/W with fast moving speed.

Code description: G28 is non-modal G code;

X, Z: the absolute coordinates of the intermediate points;

U, W: the difference between the X-axis absolute coordinates of the center point position and the starting position.

5.2.8 G50 The floating workpiece coordinate system sets G50

Code format: G50 X/U Z/W;

Code function: set the absolute coordinates of the current position, and set up the floating workpiece coordinate system in the system by setting the absolute coordinates of the current position. After executing this code, the system returns the current location as the zero point of the program and returns the zero operation of the program. After the floating workpiece coordinate system is established, the absolute coordinate program is programmed to input the coordinate values in this coordinate system until the G50 is executed again to establish a new working coordinate system.

Code description: G50 is non-modal G code;

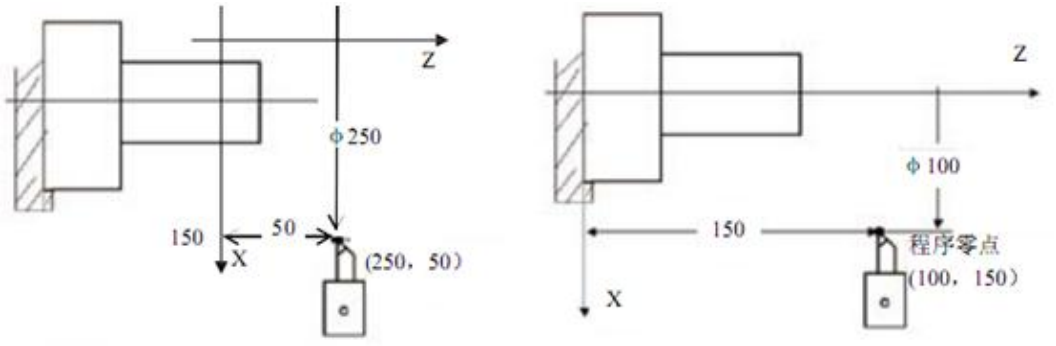
X: the current position of the new X-axis absolute coordinates;

U: the difference between the absolute coordinate of the new X-axis and the absolute coordinate before the code is executed.

Z: the current position of the new Z axis absolute coordinates;

W: the difference between the absolute coordinate of the new Z axis and the absolute coordinate before the code is executed.

In the G50 code, the X/U, Z/W unentered, does not change the current position value, setting the current position value to the zero of the program (when the G50 SXXXX is not set zero).



Before using G50 Set the coordinate **After using G50 Set the coordinate**

3-16

As shown in figure 3-16, when the code snippet is "G50 x 100 Z150;" After that, the workpiece coordinate system is set up as shown in the figure, and the (X100Z150) point is set to program zero.

5.2.9 Workpiece coordinate system G54 ～ G59

Code format: G54 ~ G59

Code function: to specify the current workpiece coordinate system, select the workpiece coordinate system by specifying the method of the workpiece coordinate G code in the program.

Code description:

1. No instruction parameters.

2. The system itself can set up six workpiece coordinate systems, which can be selected by the instruction G54 ~ G59

G54…….. Workpiece coordinate system 1

G55…….. Workpiece coordinate system 2

G56…….. Workpiece coordinate system 3

G57…….. Workpiece coordinate system 4

G58…….. Workpiece coordinate system5

G59…….. Workpiece coordinate system 6

3. When different coordinate systems are called in the program segment, the axis of the instruction movement will be located to the coordinate points in the new workpiece coordinate system; Without the moving axis of instruction, the coordinates will jump to the corresponding coordinate values in the new workpiece coordinate system, and the actual machine position will not change.

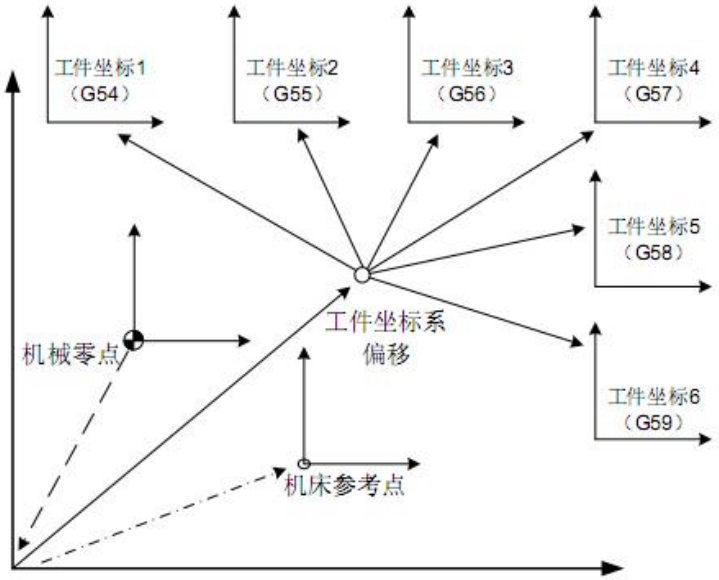
Example: the coordinate system of the frame origin of G54 corresponds to (20, 20).

The coordinate system of the coordinate system of G55 is (30, 30).

When the sequence executes, the absolute coordinates of the endpoint are shown as follows:

表 3.11.1

程序	绝对坐标	机床坐标
G0 G54 X50 Z50	50，50	70，70，70
G55 X100	100，40	130，70
X120 Z80	120，80	150，110

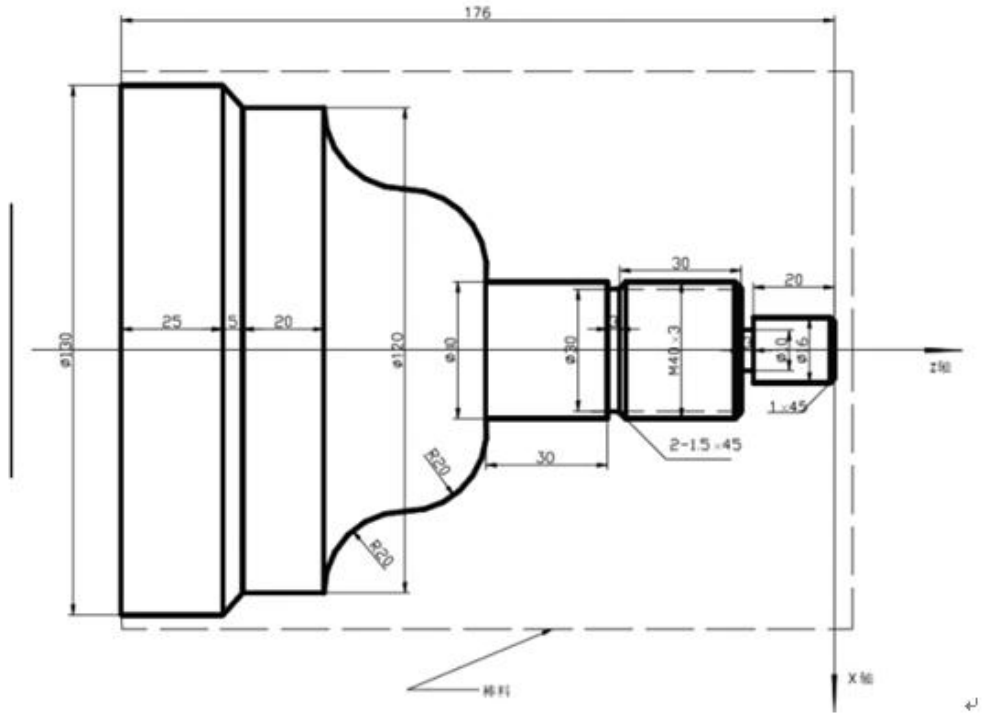


As shown in the figure above, the machine tool will return to the machine zero manually after the machine is turned on, and the machine coordinate system will be established by the machine zero, which will generate the reference point of the machine tool and determine the coordinate system of the workpiece. The corresponding value of the external workpiece origin offset data parameter P270 ~ 274 is the total offset of 6 workpiece coordinate systems. Can entry mode input data or set parameters of coordinate offset P128 ~ P139, P275 ~ 292 can specify six workpiece coordinate system origin, the six workpiece coordinate system is based on the distance from the mechanical zero zero point to the respective coordinate system and the setting.

Chapter 6 programming

6.1 programming

Shown below workpieces, the size of the bar for Φ 136 mm x 180 mm



1号刀：	外圆粗车刀	
2号刀	外圆精车刀	
3号刀	切槽刀	3mm
4号刀	螺纹刀	60度公制螺纹刀

programming

According to the machining process and the code interpretation of this manual, set up the workpiece coordinate system shown in figure 14-1, and the editing procedure is as follows

O0001; Part name

N0000 G0 x 150 Z50; Locate to safe position for knife

N0005 M12. Clamp chuck (if the machine is equipped with automatic clamping device such as hydraulic chuck)

N0010 M3 S800; Open spindle, speed 800

N0020 M8. A coolant

N0030 T0101; First knife

N0040 G0 x 136 Z2; Close to the workpiece

N0050 G71 U0.5 R0.5 F200; Cut the depth 1mm and return the knife 1mm

N0055 G71 P0060 Q0150 U0.25 W0.5; The X axis is reserved 0.5mm, and the z-axis is 0.5mm

N0060 G0 X16; Close to the workpiece end face

N0070 G1 Z - 23. Car Φ 16 cylindrical

N0080 X39.98; surfacing

N0090 W - 33; Car Φ cylindrical 39.98

N0100 X40; surfacing

N0105 W - 30; Car Φ 40 cylindrical

N0110 G3 x 80 W-20 R20; Car convex arc
N0120 G2 x 120 W-20 R20; Car concave arc
N0130 G1 W - 20; Car Φ 120 cylindrical
N0140 G1 X130 w-5; tapering
N0150 G1 W - 25; Car Φ 130 cylindrical
N0160 G0 x 150 Z185; Return to the knife point when the cart is finished
N0170 T0202; Take knife no. 2, and perform knife no. 2
N0180 G70 P0060 Q0150; Fine car circulation
N0190 G0 x 150 Z185; Return to the knife point when the cart is finished
N0200 T0303; Change number 3 knife, execute 3 knife deviation
N0210 G0 z-56 X42; Close to the workpiece
N0220 G1 x 30 F100; 30 slot cut Φ
N0230 G1 X37 F300; return
N0240 G1 X40 W1.5; Chamfer Angle
N0250 G0 x 42 W30; Give out the cutting knife width
N0260 G1 X40;
N0262 G1 X37 W1.5; Chamfer Angle
N0264 G1 X10. Cut Φ 10 slot
N0266 G0 x 17 Z minus 1;
N0268 G1 X16;
N0270 G1 X14 Z0 F200; Chamfer Angle
N0280 G0 x 150 Z50; Return change point
N0290 T0404 according to; Change the number 4, set the spindle 100 speed
N0300 G0 x 42 Z minus 20; Close to the workpiece
N0310 G92 X39 w-34 F3; Cutting thread circulation (omitted J, K tail control)
N0320 X38; Enter the second knife for 1mm
N0320 X37; Feed 1mm and then cut the third knife
N0330 X36.4; Enter 0.6mm to cut the fourth knife
N0332 X36; Enter the fifth knife for 0.4mm
N0340 G0 x 150 Z50; Back in the knife point
N0350 T0100 U0 W0; Switch back to the 1 knife and execute the offset value
N0360 M5;Close the main shaft
N0370 M9;Close the cooling fluid
N0380 M13;Release the chuck (if the machine is mounted with automatic clamping device such as hydraulic chuck)
N0390 M30 ;// program completion;

6.2 The M code table defined by the root 1000TC standard PLC

代码	功能	备注
M00	程序暂停	
M01	程序选择停	
M03	主轴逆时针转	功能互锁，状态保持
M04	主轴顺时针转	
*M05	主轴停止	
M08	冷却液开	功能互锁，状态保持
*M09	冷却液关	
M10	尾座进	功能互锁，状态保持
M11	尾座退	
M12	卡盘夹紧	功能互锁，状态保持
M13	卡盘松开	
M14	主轴位置控制	功能互锁，状态保持
*M15	主轴速度控制	
M20	主轴夹紧	功能互锁，状态保持
*M21	主轴松开	
M24	第2主轴位置控制	功能互锁，状态保持
*M25	第2主轴速度控制	
M32	润滑开	功能互锁，状态保持
*M33	润滑关	
M63	第2主轴逆时针转	功能互锁，状态保持
M64	第2主轴顺时针转	
*M65	第2主轴停止	
*M41、M42、M43、M44	主轴自动换档	功能互锁，状态保持

6.3 G Code table

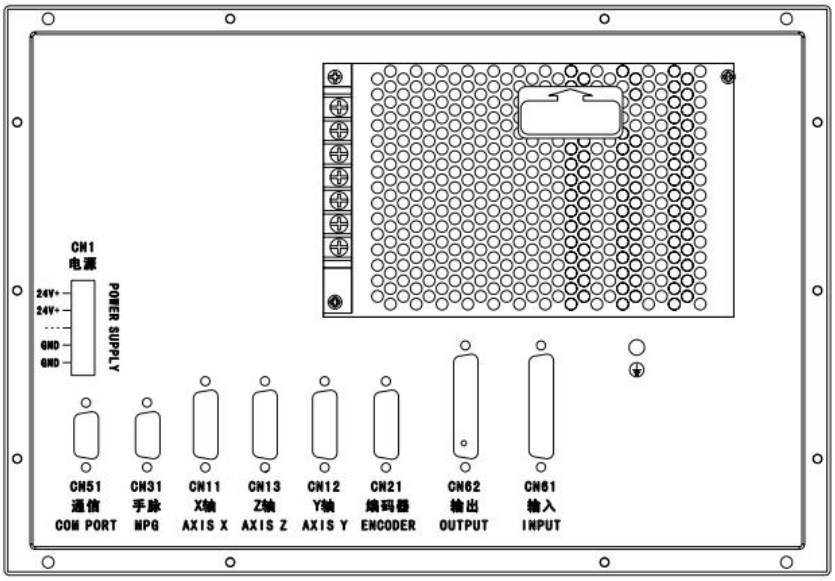
G List of code words

指令字	组别	功能	备注
G00	01	快速移动	初态 G 代码
G01		直线插补	模态 G 代码
G02		圆弧插补（顺时针）	
G03		圆弧插补（逆时针）	
G32		螺纹切削	
G33		Z 轴攻丝循环	
G34		变螺距螺纹切削	
G90		轴向切削循环	
G92		螺纹切削循环	
G94		径向切削循环	
G04	00	暂停、准停	非模态 G 代码
G10		数据输入方式有效	
G11		取消数据输入方式	
G12		存储行程检测功能接通	

指令字	组别	功能	备注
G13	00	存储行程检测功能断开	非模态 G 代码
G27		返回参考点检测	
G28		返回机床第 1 参考点	
G29		从参考点自动返回	
G30		返回机床第 2、3、4 参考点	
G31		跳转插补	
G50		坐标系设定	
G65		宏代码	
G70		精加工循环	
G71		轴向粗车循环	
G72		径向粗车循环	
G73		封闭切削循环	
G74		轴向切槽多重循环	
G75		径向切槽多重循环	
G76		多重螺纹切削循环	
G54	05	工件坐标系 1	模态 G 代码
G55		工件坐标系 2	
G56		工件坐标系 3	
G57		工件坐标系 4	
G58		工件坐标系 5	
G59		工件坐标系 6	
G20	06	英制单位选择	模态 G 代码
G21		公制单位选择	
G96	02	恒线速开	模态 G 代码
G97		恒线速关	初态 G 代码
G98	03	每分进给	初态 G 代码
G99		每转进给	模态 G 代码
G40	07	取消刀尖半径补偿	初态 G 代码
G41		刀尖半径左补偿	模态 G 代码
G42		刀尖半径右补偿	
G17	16	XY 平面	模态 G 代码
G18		ZX 平面	初态 G 代码
G19		YZ 平面	模态 G 代码

Chapter 7 the panel Install and nection

7.1 1000TC Rear cover interface layout

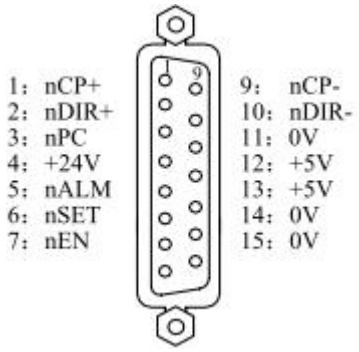


7.2 Interface specification

◎ Power box: provide + 24V, GND power supply
◎ CN1: power interface
◎ CN11: X axis, 15 core D socket, connect the X-axis drive unit
◎ CN12: Y-axis, 15 core D socket, connect the Y-axis drive unit
◎ CN13: Z axis, 15 core D socket, connect z-axis drive unit
◎ CN14: A shaft, 15 core D socket, connect A shaft drive unit
◎ CN21: encoder, 15 core D pin socket, connect the main shaft encoder
◎ CN31: handwheel, 26 core D pin socket, connected handwheel
◎ CN51: communication, 9 core D socket, connect PC RS232 interface
◎ CN61: input, 25 core D pin socket, connect machine input
◎ CN62: output, 25 core D socket, connect machine output

7.3 Interface signal definition

7.3.1 Drive signal interface



信号	说明
nCP+、nCP-	指令脉冲信号
nDIR+、nDIR-	指令方向信号
nPC	零点信号
nALM	驱动器报警信号
nEN	轴使能信号
nSET	脉冲禁止信号

图 2-1 CN11、CN12、CN13、
(15Core D socket socket) interface

7.3.2 Instruction pulse signal and instruction direction signal

NCP +, nCP- for command pulse signal, nDIR +, nDIR- for instruction direction signal, both sets of signals are in the differential (AM26LS31) output external recommendation using AM26LS32 reception, internal circuits are shown in figure 2-2 below:

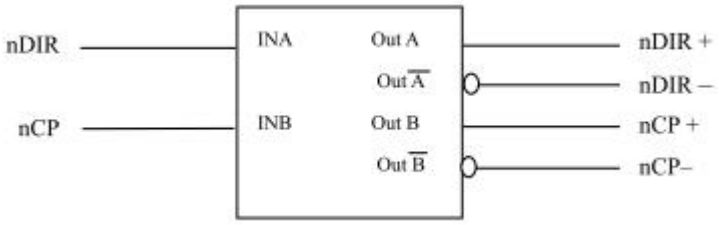


Figure 2-2 the internal circuit of the pulse signal and the direction signal

7.3.3 nALM Drive unit alarm signal nALM

By the CNC parameter №. 009 Bit0, Bit1, Bit2 set drive unit alarm level is low level or high level. The internal circuit is shown in figure 2-3:

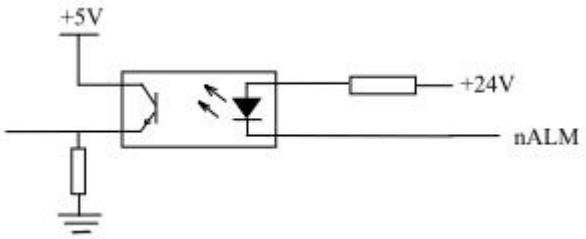


Figure 2-3 internal circuit of driver alarm signal

7.3.4 nEN The shaft allows the energy to signal nEN

When CNC works normally, nEN signal output is effective (nEN signal is connected with 0V). When the driver unit is alarm, CNC shuts off nEN signal output (nEN signal is disconnected from 0V). The internal interface circuit is shown in figure 2-5 below:

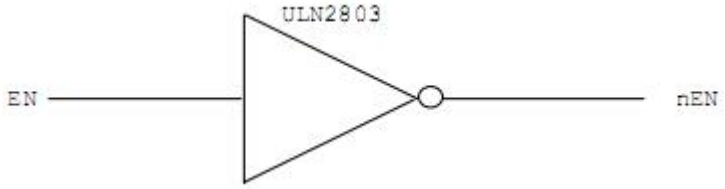


Figure 2-5 axis enables the internal interface circuit

(b) the connection method of using an NPN type hall element for both deceleration signal and zero signal signal is shown in figure 2-9 below:

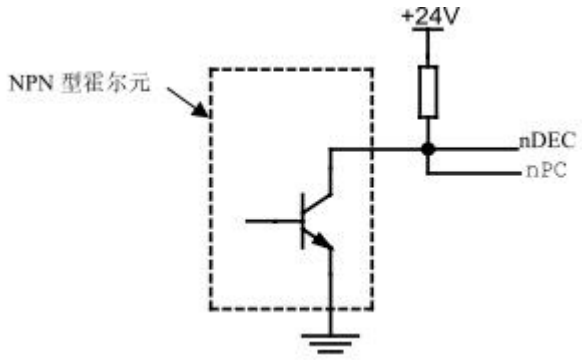


Figure 2-9 connect the NPN type hall elements

(c) the connection method of a PNP type hall element for both deceleration signal and zero signal is shown in figure 2-10:

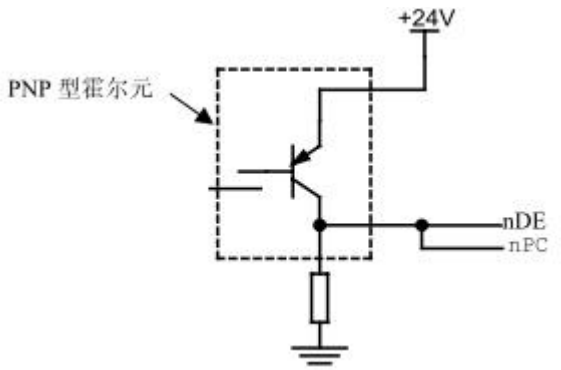
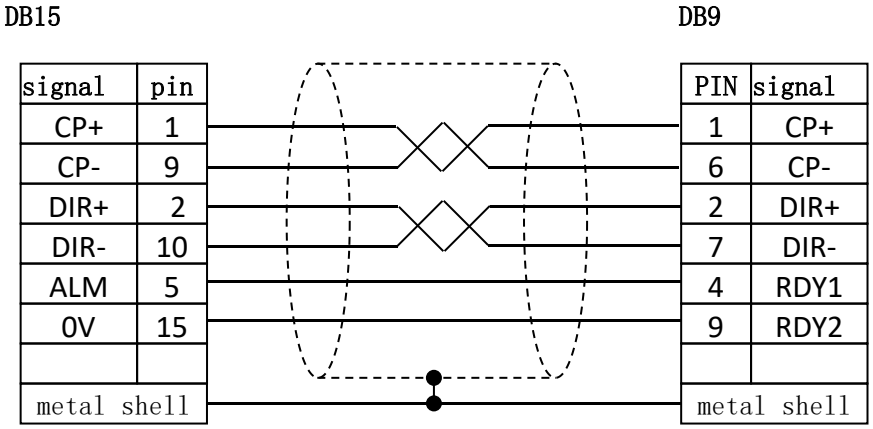


Figure 2-10 connect the PNP type hall elements

7.3.5 Connection with the driver unit

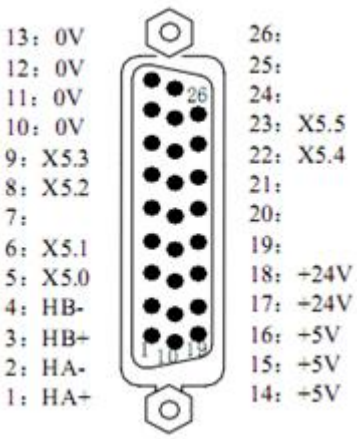
The connection between 1000TC and drive unit is shown in figure 2-11

1000TC（CN11、 CN12、CN13） 3HE driver unit signal interface



7.4 Connection with the handwheel

7.4.1 Handwheel interface definition

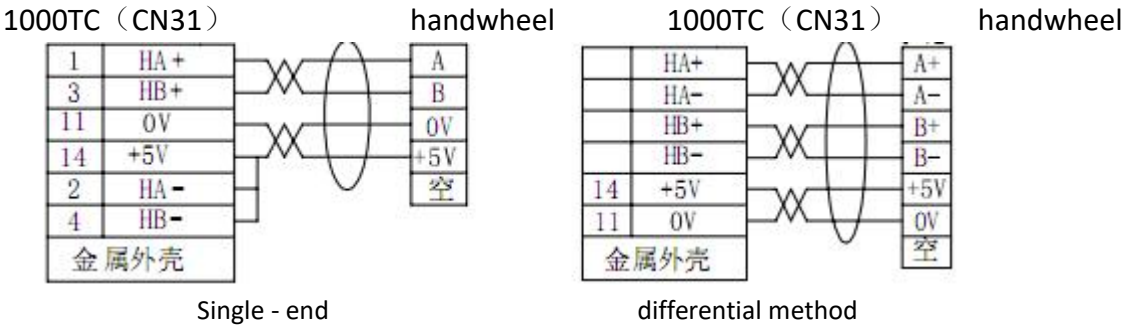


信号	说明
HA+、HA-	手轮 A 相信号
HB+、HB-	手轮 B 相信号
X5.0	X手轮轴选
X5.1	Y手轮轴选
X5.2	Z手轮轴选
X5.3	增量×1
X5.4	增量×10
X5.5	增量×100
+24V	直流电源
VCC、GND	

FIG. 2-5 CN31 handwheel interface
(26 core hole DB socket)

7.4.2 Signal introduce

The connection between 1000TC and handwheel is shown in figure 2-17:



2-17

7.5 Common frequency converter connection

Analog main shaft interface SVC end can output 0 ~ 10V voltage

The connection between 1000TC and inverter is shown in figure 2-21:

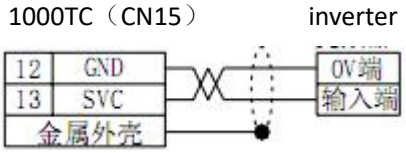


图 2-21 1000TC with inverter

7.6 Power interface connection

At 1000TC, the power box is connected to the CN1 interface of 1000TC. The user only needs 220V ac power.

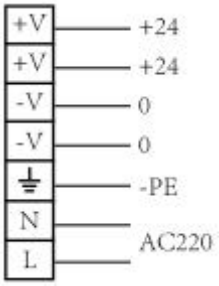


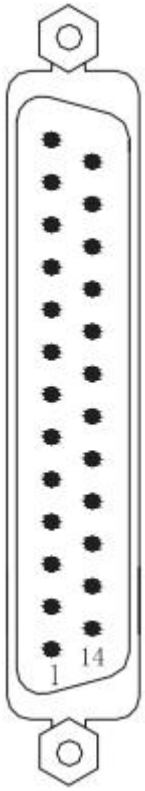
Figure 2-24 system power interface CN1

7.7 I/O interface definition

Pay attention!

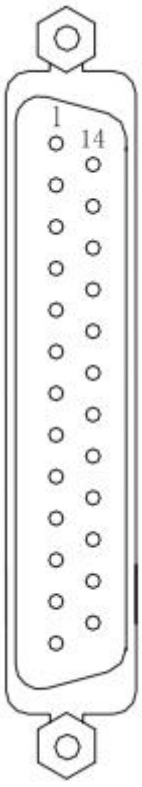
1000 tc CNC lathe with fixed address I/O functions Defined by the PLC program (ladder diagram), when 1000 tc CNC lathe assembly of machine tool, the I/O function is determined by the machine manufacturer to design, specific please refer to the machine manufacturer's instructions.

The I/O function of the fixed address is not marked in this section, which is described in the standard PLC program of 1000TC. Not specifically said Attention, please!



input interface（CN61）

PIN	addr	function	description
3,10, 19	0V	Power interface	Power 0V
11,23, 16 7,20,4	+24V	Power interface	Power+24V
1	X0.0	JS	Count function
14	X0.1	SP	Out pause
2	X0.2	DIQP	Chuck control input
15	X0.3	DECX	X axis speed dec
17	X0.4	DITW	Tailstock control
5	X0.5	ESP	Sunden stop
18	X0.6	LMIY	Y axis out of range
6	X0.7	T01	Knife positon 1
8	X1.0	T02	Knife positon2
21	X1.1	T03	Knife positon 3
9	X1.2	T04	Knife positon 14
22	X1.3	DECZ	Z axis speed dec
24	X1.4	ST	External loop start
12	X1.5	LMIX	X axis out of range
25	X1.6	LMIZ	Z axis out of range
13	X1.7	DECY	Y axis speed dec



output interface（CN62）

PIN	ADDR	FUNCTION	Description
3,16,19 7,10,23	0V	Power	Power 0V
4,20,11	+24V	Power	Power＋24V
1	Y0.0	M08	Mill spindle output
14	Y0.1	M32	The tail cap output
2	Y0.2	SCLP	The spindle clamping
15	Y0.3	M03	Spindle rotation CCW
17	Y0.4	M04	Spindle rotation CW
5	Y0.5	M05	Spindle stop
18	Y0.6	cylinder 1	M70 OUTPUT
6	Y0.7	cylinder 2	M72 OUTPUT
8	Y1.0	M80/M81 L0	Custom OUTPUT 1
21	Y1.1	M80/M81 L1	Custom OUTPUT 2
9	Y1.2	M80/M81 L2	Custom OUTPUT 3
22	Y1.3	M80/M81 L3	Custom OUTPUT 4
24	Y1.4	气缸 3	M74 OUTPUT
12	Y1.5	气缸 4	M76 OUTPUT
25	Y1.6	TL+	Tool carriage forward
13	Y1.7	TL -	Tool reversal

Note 1: partial input and output interface can define many functions, in the above table, "/";
Note 2: when the output function is valid, the output signal and the +24V guide. When the output function is invalid, the output signal is high impedance cut-off.
Note 3: input signal with +24V guide, this input is valid. Input signal and +24V cut-off, the input is invalid;
Note 4: + 24V, COM is equivalent to the terminal terminal of the 1000TC auxiliary power box;

7.7.1 Input signal

Input signal refers to the signal from machine tool to CNC, when the input signal is connected with + 24V, the input is valid; When the input signal is disconnected with + 24V, the input is invalid. The input signal should meet the following conditions on the machine side of the contact:
Contact capacity: DC30V, 16mA above
Leakage current at the contact point at the opening: 1mA below
Voltage drop between contacts during access: below 2V (current 8.5 mA, including voltage drop of cable)
Input signal of the external input there are two ways: use a contact switch input, in this way the signal from the side of the machine tool buttons, limit switch and relay contact, connection as shown in figure 2-27:

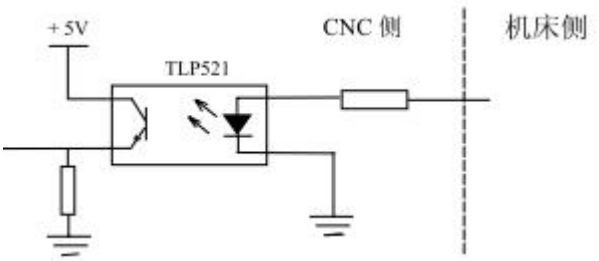


figure **2-27**

Another USES the contactless switch (transistor) input, and the connection is shown in figure. 2-28a and figure 2-28b.

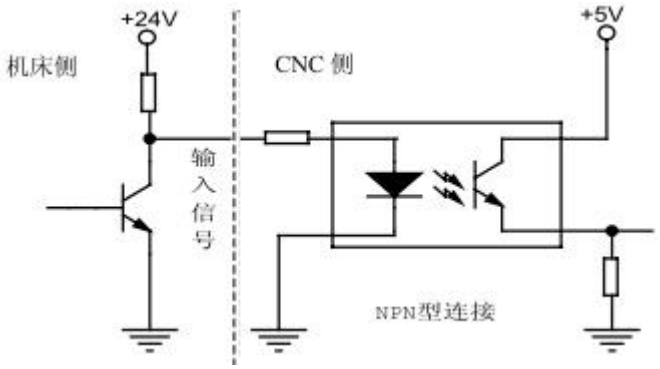


figure **2-28A NPN** connection

7.7.2 output signal

The output signal is used to drive the relay and indicator light on the machine side. The output function is effective when the output signal is connected with the + 24V. When the +24V is disconnected, the output function is invalid. The I/O interface has a 16-way digital output, all of which have the same structure, as shown in figure 2-29:

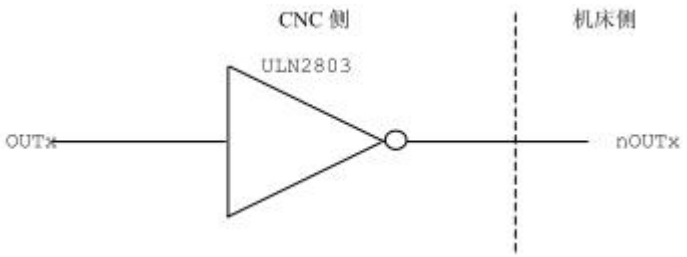


figure **2-29 Digital output module circuit structure diagram**

The logical signals OUTx output by the motherboard are sent by the connector to the input side of the inverter (ULN2803). NOUTx has two output states: 0V output or high resistance. Typical applications are as follows:

Drive the light-emitting diode

Using the ULN2803 output driven light-emitting diode requires a series of resistances to restrict the current of the light-emitting diode (generally about 10mA). As shown in figure 2-30 below:

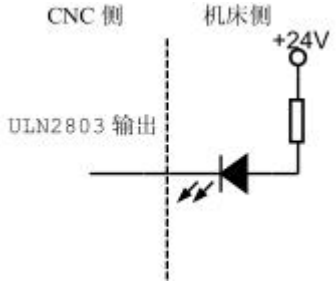


figure **2-30**

Drive lamp type indicator light

Using the ULN2803 output drive filament type indicator, a preheating resistance is required to reduce the current shock on the guide, and the preheating resistance resistances to make the indicator light not bright as the principle, as shown in figure 2-31.

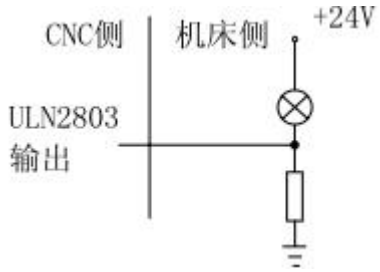


figure **2-31**

Driving inductive load (such as relay)

Using the ULN2803 output to drive the perceptual load, a continuation diode is needed near the coil to protect the output circuit and reduce interference. As shown in figure 2-32.

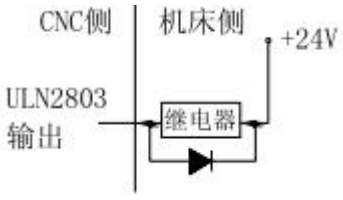


figure **2-32**

7.8 I/O functions and connections

7.8.1 Travel limit and stop

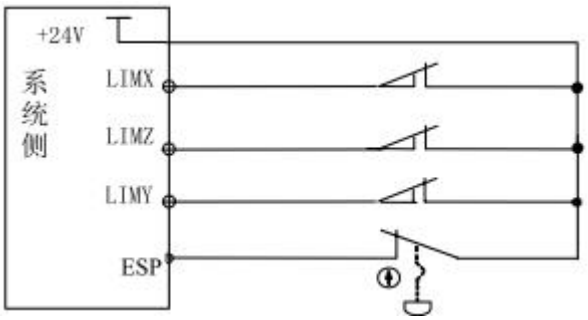


Figure 2-34b emergency stop, travel switch independent connection

Description:

The limit of travel is connected with a quick stop

When there is a distance or press the stop button, the CNC can appear "scram" alarm, such as distance, the distance to lift button not loosen, press the reset button to cancel the alarm after the distance may be discharged to move in the opposite direction. When emergency stop alarm, CNC stop pulse output. Besides the function of the above CNC, the other functions can be defined by PLC program when the alarm is stopped. The standard PLC program defines the function as: to close the M03 or M04, M08 signal output, and output the M05 signal when the alarm is stopped.

The travel limit is connected to the emergency stop

1. There is only one hyperrange contact for each axis, and the positive and negative hyperrange alarm is judged by the direction of the axis.

2. When the overpass alarm occurs, it can be moved in the opposite direction, and after removing the limit position, the alarm can be removed according to the reset.

Note: before the super - limit function is enabled, it is necessary to ensure that the machine tool is in a positive and negative stroke, otherwise the alarm will not be true.

Chapter 8 debugging of machine tools

This chapter introduces the test operation method and its steps of 1000TC for the first time, and the corresponding machine tool operation can be carried out according to the following operation steps.

8.1 Quick stop and limit

1000TC has the software limit function. For the sake of safety, it is recommended that the hardware limit measures should be adopted at the same time. In the positive and negative direction of each axis, the limit switch of the travel limit is installed, and the connection is shown in Figure. 2-34b

8.2 Driver unit Settings

According to the set alarm logic level state parameters of drive unit of № 009 BIT2, BIT1, BIT0 (YALM, ZALM, XALM, corresponding to the Y, Z, X axis), supporting the company drive unit time state parameters of № 009 BIT2, BIT1, BIT0 bits set to 1.

If the machine moving direction and instruction requirement direction, can modify the parameters of № 008 BIT2, BIT1 and BIT0 (DIRY, DIRZ, DIRX corresponding Y, Z, X axis).

Manually move through parameter of № 175 BIT2, BIT1 and BIT0 (YVAL ZVAL XVAL corresponding Y, Z, X axis movement keys) to change.

8.3 Gear ratio adjustment

When the distance between the moving distance of the machine tool and the displacement distance of the CNC coordinate is inconsistent, the adjustment of the electronic gear ratio can be modified to adapt the different mechanical transmission ratio.

Calculation formula:

$$\frac{CMR}{CMD} = \frac{\delta \times 360}{\alpha \times L} \times \frac{Z_M}{Z_D}$$

CMR ： Instruction times multiplication coefficient（The data parameter № 015 、 № 016 、 № 146 、 № 147 、 № 148 ）

CMD ： Frequency of instruction（he data parameter № 017 、 № 018 、 № 149 、 № 150 、 № 151 ）

α ： The pulse equivalent, the motor receives a pulse rotation Angle

L ： Lead of screw

δ ： The current input minimum unit of CNC

ZM ： The number of teeth of screw end gear

ZD ： The number of teeth of the motor end gear

Example: the tooth number of the screw end gear is 50, the tooth number of the motor end gear is 30, the pulse equivalent of alpha =0.075 degree, and the lead of the wire rod is 4 mm; X, z-axis electronic gear

ratio: $$\frac{CMR}{CMD}=\frac{\delta\times 360}{\alpha\times L}\times\frac{Z_M}{Z_D}=\frac{0.001\times 360}{0.075\times 4}\times\frac{50}{30}=\frac{2}{1}$$

Data parameter № 015 （ CMRX ）=2 ， № 017 （ CMDX ）=1 ； № 016 （ CMRZ ）=2 ， № 018 （ CMDZ ）=1 。

When the electronic gear ratio is larger than the numerator, the maximum speed of the CNC will decrease. Example: data parameters № 016 （ CMRZ ） =2 ， № 018 （ CMDZ ） =1 时， The maximum speed allowed by the Z axis is 8000mm/min 。

When the electronic gear ratio is not equal to the numerator and the denominator, the positioning accuracy of CNC may decrease. Example: data parameters № 016（ CMRZ ） =1 ， № 018 （ CMDZ ） =5 时 ， The input increment is 0.004 without the output pulse, and the input increment reaches 0.005 to output a pulse.

8.4 Back clearance compensation

The gap compensation is the input value based on the actual clearance volume. The unit is mm (metric machine) or inch (English machine). Can use a dial indicator, dial gauge or measure laser detector, backlash compensation must carry on the accurate compensation can improve the accuracy of machining, so don't recommend the handwheel or single step mode is used to measure the screw backlash, suggested as follows: to measure the reverse clearance

Edit program (Z axis for example) :

O 0001 ；

N10 G01 W10 F800 ；

N20 W15 ；

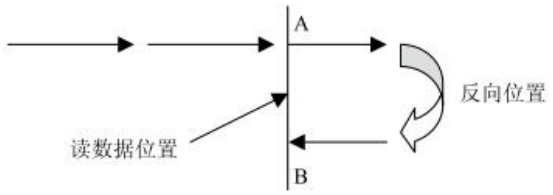
N30 W1 ；

N40 W-1 ；

N50 M30 。

The compensation value of the reverse clearance error should be set to zero before measurement.

Single run program, locate the two times, find the benchmark A, record the current data, then run the same to the 1mm, then reverse the 1mm to B, and read the current data.



Reverse clearance error compensation =|A point record data - |; Calculated the data input into the CNC parameter № 034, 035, 180, № № № 035 or № 182.

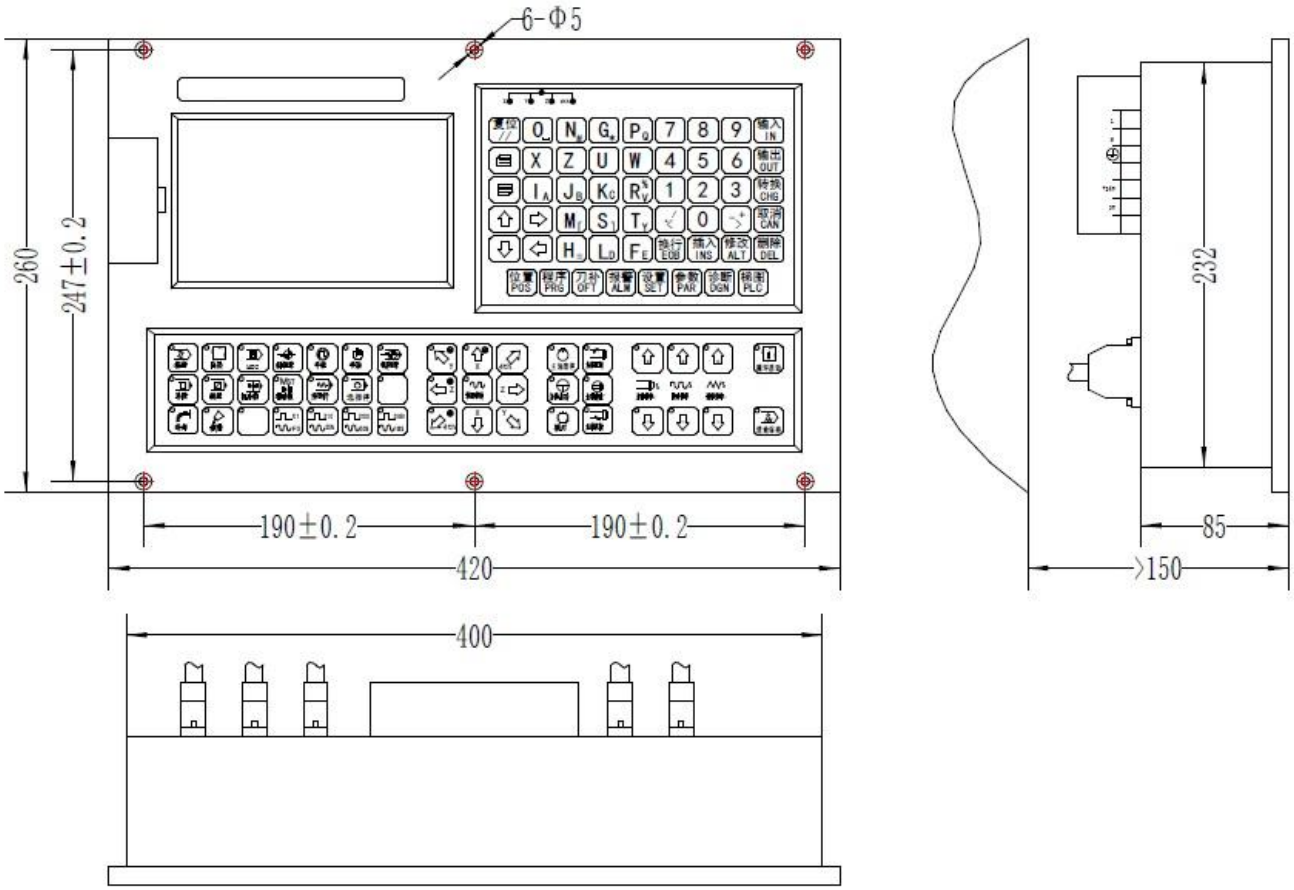
Data A: read the data of the percentage at A;

Data B: read the percentage of the data in B;

Note 1: CNC parameter of № 011 Bit7 backlash compensation can be set, the data parameter № 246 ~ № 249 backlash clearance can be set at a fixed frequency compensation compensation step length;

Note 2: the machine tool should be retested for reverse clearance after three months.

Appendix 1 1000TC dimensions



Appendix 1 1000TC dimensions