CENTER OF TECHNOLOGICAL INITIATIVES, LLC

info@cti-sumy.com.ua

Предлагаем услуги по сервисному обслуживанию и поставке оборудования Shenzhen Guanhong Automation (SZGH).
В рамках ремонта мы выполним осмотр и диагностику неисправностей, техническое обслуживание и замену повреждённых элементов, а также консультацию по установке и эксплуатации.

У нас большой опыт в работе с оборудованием SZGH. Мы часто используем его при изготовлении собственных машин

Контакты:

http://cti-sumy.com.ua
info@cti-sumy.com.ua
sales1@cti-sumy.com.ua
https://www.facebook.com/cti.sumy
https://www.youtube.com/channel/UCyYGmKVZ2FwZnoX4RV8Is0g

Машины под индивидуальные требования

Сварочный станок

Сварочный станок

Сварочный станок

Сварочный полуавтоматический станок

Сварочный станок

User Manual

SZGH-CNC1000MDcb(series)
Milling Control System
-OPERATION-
V3.2

-BOOK2-

Shenzhen Guanhong Automation CO.,LTD
Website: www.szghauto.com
Add:Room 503 Anxin Building, No 536 Shenhui Road, Liuyue community, Henggang Street ,Longgang District, Shenzhen City,Guangdong Province, China
Post code: 518115

Warnings and Notes as Used in this Publication

Warning

Warning notices are used in this publication to emphasize that hazardous voltages, currents, temperatures, or other conditions that could cause personal injury exist in this equipment or may be associated with its use.
In situations where inattention could cause either personal injury or damage to equipment, a Warning notice is used.

Caution

Caution notices are used where equipment might be damaged if care is not taken.

Note

Notes merely call attention to information that is especially significant to understanding and operating the equipment.

This document is based on information available at the time of its publication. While efforts have been made to be accurate, the information contained herein does not purport to cover all details or variations in hardware or software, nor to provide for every possible contingency in connection with installation, operation, or maintenance. Features may be described herein which are not present in all hardware and software systems. SZGH Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

Shenzhen Guanhong Automation makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. No warranties of merchant-ability or fitness for purpose shall apply.

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units. It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.

Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

1 Definition of Warning , Caution, and Note

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into Warning and Caution according to their bearing on safety. Also, supplementary information is described as a Note. Read the Warning, Caution, and Note thoroughly before attempting to use the machine.

WARNING

Applied when there is a danger of the user being injured or when there is a danger of both the user being injured and the equipment being damaged if the approved procedure is not observed.

CAUTION

Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

NOTE

The Note is used to indicate supplementary information other than Warning and Caution.

Read this manual carefully, and store it in a safe place.

2 GENERAL WARNINGS AND CAUTIONS

Warning

1. Never attempt to machine a workpiece without first checking the operation of the machine.
Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
2. Before operating the machine, thoroughly check the entered data. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
4. When using a tool compensation function, thoroughly check the direction and amount of Compensation. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
5. The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change. Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
6. Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.
7. The operator's manual and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.
8. Some functions may have been implemented at the request of the machine–tool builder. When using such functions, refer to the manual supplied by the machine–tool builder for details of their use and any related cautions.

NOTE: Programs,parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off. Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied operator's manual and programming manual carefully such that you are fully familiar with their contents.

Warning

1.Coordinate system setting

If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command.

Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2. Positioning by nonlinear interpolation

When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming. Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Function involving a rotation axis

When programming polar coordinate interpolation or normal–direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely. Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4. Inch/metric conversion

Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position. Before starting the machine, therefore, determine which measurement units are being used. Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

5. Constant surface speed control

When an axis subject to constant surface speed control approaches the origin of the workpiece coordinate system, the spindle speed may become excessively high. Therefore, it is necessary to specify a maximum allowable speed. Specifying the maximum allowable speed incorrectly may damage the tool, the machine itself, the workpiece, or cause injury to the user.

6. Stroke check

After switching on the power, perform a manual reference position return as required. Stroke check is not possible before manual reference position return is performed. Note that when stroke check is disabled, an alarm is not issued even if a stroke limit is exceeded, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

7. Absolute/incremental mode

If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

8. Plane selection

If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

9. Compensation function

If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine. Before issuing any of the above commands, therefore, always cancel compensation function mode.

4 WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied operator's manual and programming manual carefully, such that you are fully familiar with their contents.

Warning

1. Manual operation

When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2. Manual reference position return

After switching on the power, perform manual reference position return as required. If the machine is operated without first performing manual reference position return, it may behave
unexpectedly. Stroke check is not possible before manual reference position return is performed.
An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Manual handle feed

In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

4. Disabled override

If override is disabled (according to the specification in a macro variable) during threading or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

5. Origin/preset operation

Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.

6. Workpiece coordinate system shift

Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully. If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

7. Software operator's panel and menu switches

Using the software operator’ s panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator’ s panel, such as mode change, override value change, and jog feed commands.
Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

8. Manual intervention

If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

9. Feed hold, override, and single block

The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

10. Dry run

Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

11. Cutter and tool nose radius compensation in MDI mode

Pay careful attention to a tool path specified by a command in MDI mode, because tool nose radius compensation is not applied. When a command is entered from the MDI to interrupt in automatic operation in tool nose radius compensation mode, pay particular attention to the tool path when automatic operation is subsequently resumed. Refer to the descriptions of the corresponding functions for details.

12. Program editing

If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.

5 WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1. Memory backup battery replacement

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high－voltage circuits (marked and fitted with an insulating cover).Touching the uncovered high－voltage circuits presents an extremely dangerous electric shock hazard.

NOTE:The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied. If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen. When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC memory will be lost. Refer to the maintenance section of the operator's manual for details of the battery replacement procedure.

2. Absolute pulse coder battery replacement

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on,and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work. When replacing the batteries, be careful not to touch the high－voltage circuits (marked and fitted with an insulating cover). Touching the uncovered high－voltage circuits presents an extremely dangerous electric shock hazard.
NOTE:The absolute pulse coder uses batteries to preserve its absolute position. If the battery voltage drops, a

low battery voltage alarm is displayed on the machine operator's panel or screen. When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.

3. Fuse replacement

For some units, the chapter covering daily maintenance in the operator’ s manual or programming manual describes the fuse replacement procedure.

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse. For this reason, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing a fuse with the cabinet open, be careful not to touch the high－voltage circuits(marked and fitted with an insulating cover).

Touching an uncovered high－voltage circuit presents an extremely dangerous electric shock hazard.

CONTENTS

Chapter 1 Operation

When using CNC Milling system, just master the parameter of system, edit program, manual operation, auto operation. Then you can operate the system easily. There are some details to instruct hereinafter.

1.1 Operational Panel

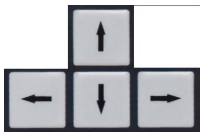
This system panel is total controller ,which includes 8.4 inches LCD display area , function menu , editing keyboard area & machine control panel.(take E panel as example)

Fig1.1 SZGH-CNC1000MDc CNC Milling Controller

1.2 Function Menu

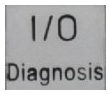
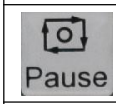
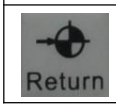
Menu Keys	Comment	
Parameter	Enter the interfaces of status parameter. Dates parameter , diagnosis and screw compensation parameter interface (interfaces can be switched by repeated press)	
Program	Enter the program interface.	
Redeem	Enter the redeem interface.	
Manual	Enter controlling condition of manual	
Auto	Enter controlling condition of auto	
Del	Delete key	***Note: Clear number of workpiece***
Esc	Exit key	
Enter	Enter key	

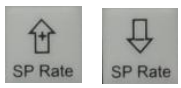
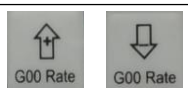
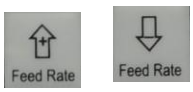
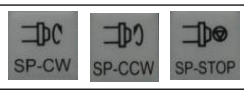
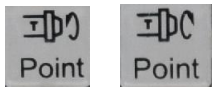
1.3 Editing Keyboard

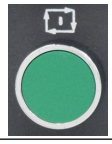
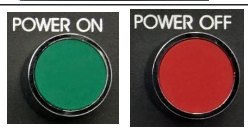
Keys	Name	Description	
	Reset key	CNC reset, stop of the feeding and moving, etc.	
	Address Key	Address input, Double-address key, switch between addresses	
	Digital Key&Symbol key	Digit input & Symbol Input	
	Cursor Move Keys	Move the cursor in different directions.	
	Page up/down key	Page up/down on display	***Note: Exchange of coordinate***
	Shift key	Shift function of key	
	Space key	Leave a blank space	

1.4 Machine Control Panel

Key	Designation	Explanation	Remarks and operation explanation
	K1 switch	K1 ON/OFF	Control Y24 ON/OFF Output Point: PIN11 of CN16 plug
	K2 switch	K2 ON/OFF	Control Y25 ON/OFF Output Point: PIN12 of CN16 plug
	K3 switch	K2 ON/OFF	Control Y26 ON/OFF Output Point: PIN13 of CN16 plug
	Coolant switch	Coolant ON/OFF	Control code: M08/M09 Output Point: PIN8 of CN3 plug

	Chuck switch	Tighten/Loosen Tool of Spindle	Control code: M10/M11 Output Point:PIN21 of CN3 plug
	Tailstock switch	Tailstock Forward/Backward	Control code: M79/M78 Output Point: PIN22 of CN3 plug
	Huff switch	Huff ON/OFF	Control code: M59/M58 Output Point: PIN6 of CN10 plug
	Single block key	To enter single block mode	
	Per Step mode key	To enter single step mode	Switching cycle from “manual continuous” to “manual increment”
	Graphic mode key	To enter graphic mode(Only Work on AUTO condition)	
	Simulate switch/dry run switch		
	Handwheel/*1/*10/ *100	Enter controlling condition of handwheel, press again, it will shift handwheel rate of *1/*10/*100	
	Diagnosis key	Enter the interfaces of diagnosis	
	Pause key	Halt for program	
	Cycle start key	Press this key and the system mode automatically runs	Auto mode, MDI mode, DNC mode
	Return Home of Machine key	To enter mode of return home of machine	

Key	Designation	Explanation	Remarks and operation explanation
	Spindle override keys	Up/Down rate of SP_Speed	Range is form 5% to 150%,16 gears totally
	Rate of G00 speed	Up/Down rate of G00_Speed	range is from 5% to 100%,16 gears totally
	Rate of feeding speed	Up/Down rate of Feeding Speed	range is from 0% to 150%,16 gears totally
	Spindle control keys	CW/CCW/Stop of Spindle	Corresponding to M03/M04/M05 codes
	Rotation of SP in JOG	Point CW/CCW rotation of spindle	

	Rotation of tool in JOG	Tool magazine rotate CW/CCW, stop until detect T08 (count of tool)
	Rapid mode	Holding Rapid key+ Manual Feeding Key, for feeding with rapid speed manually. When P38=8 in Other parameter,the key is set to switch of Rapid/Normal.
	Manual Feeding Key	For positive/negative movement of X, Y, Z, A & B axes in MANUAL mode
	Cycle Start Button, for run processing program on Auto, or run MDI	
	Pause Button, for halt processing ***Note:only E type operational panel has Pause button.***	
	Emergency stop Driver and motor stop immediately, turns off the spindle, coolant, waits for the rise of button, and initializes values	
	Interface Switch Left side: Normal Mode Middle: F-Halt , stop feeding Right side: FS-Halt, stop feeding & rotation of spindle	
	Handwheel on panel for feeding manually. P1=0 in Other parameter is select handwheel on panel. ***Note: only A type & C type operational panel have handwheel(MPG).***	
	Power On/Power OFF switches ***Note:Only C type & E type operational panel have these switches.*** ***Using solution , please check 1.7.1.4_book2***	

Note: 1."Manual speed controlled": Press "1" "2" "3" "4" "5" "6" "7" "8" "9" to set feed override "F30" "F60" "F120" "F250" "F500" "F1000" "F1500" "F2000" "F2500" "F3000".

2."Incremental coordinate": Press "Setup" to set or set to 0 after enter "relative" coordinate mode.

3. Machine coordinate clear: Press "E" in parameter and then press "Enter".

4. "F": Taking mm/min as the unit to set the manual feed speed, the input range is from 1 to 30000mm/min. And the input method according to data input method in parameter.

5."S": Set the speed of the first spindle. The range is from 0 to 99999, the max depends on the No.36 parameter in speed parameter.

6."I": Modify the increment per step in manual increment mode

7. "T": Choosing the position cutting tool.

The system adjusts one-level menu operation, intuitive, convenient, shortcut, prompt comprehensive information.Powering on system is to enter the main screen.

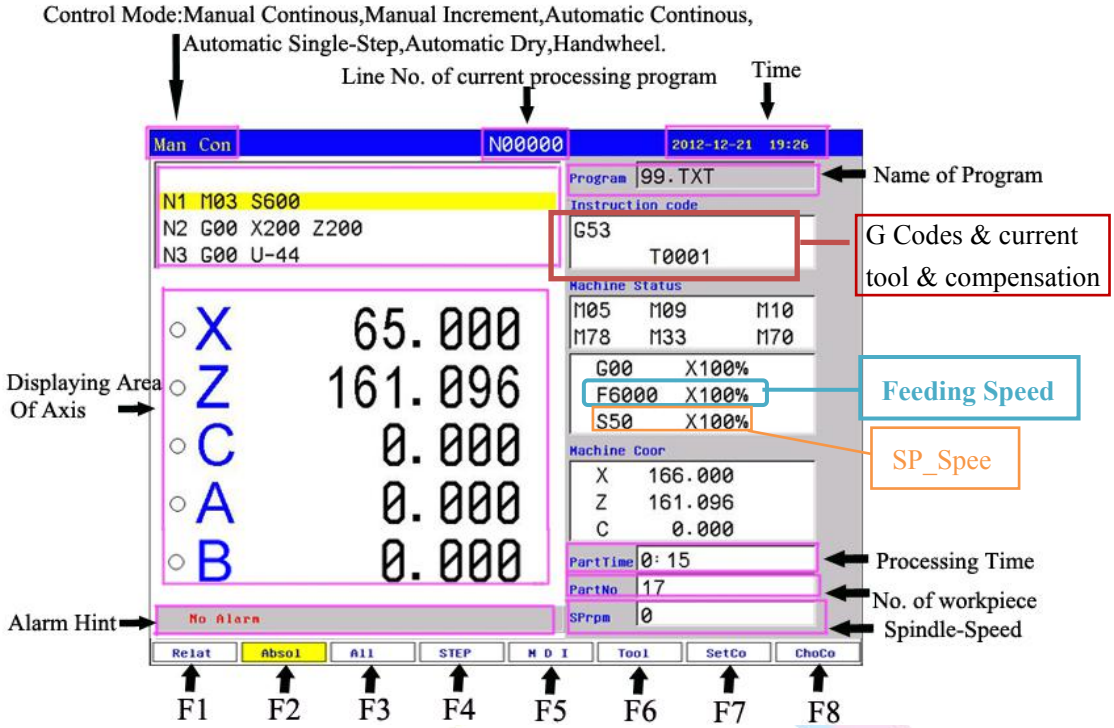


Fig1.2 Displayer

Press “Program” key enter program management area.it could edit,alter, diagnosis, delete,and copy etc.

Press “Parameter” key enter parameter management area.it could refer,alter to every parameters.

In manual condition,the system could process workpiece.

1.5 Manual Operation

1.5.1 Manual Continuous

Continuous operation is basis on the time of pressing, Press to feed, up to stop feeding. Making sure the axis and using “Manual Feeding Key” to feed, feeding speed displays on the interface (F) multi Rate.

When continuous starting, press “Rapid” to switch the speed set by P1,P2 in Speed parameter, also G00 speed. If set the speed higher than the speed in parameter, the feed speed will be P1,P2 in Speed parameter times rapid override.

In order to facilitate the user single axis cutting in the manual function, setting the manual speed in manual status. Press “F” and input the speed.

When the hard limit point beyond positive and negative feed running axis two direction at, stop the feed and prompt to feed reverse direction.(the same as hereinafter)

The manual maximum speed is limited by P3 in Speed parameter, when setting the speed is higher than the value of parameter, then will be P3 in Speed parameter.

When P38=8 in other parameter, “Rapid” is change into a switch, press once to turn on (no more to always press), press again to turn off.

1.5.2 Manual Increment

This operation is to set the value of increment as the basis, press "↑ ↓ ← →" once to run a value of increment. It will prompts "I=0010.000" in manual increment represent for the value of increment is 10mm, press "I" to revise and Enter.

The speed is the speed on display(F) times the rate.

1.5.3 Manual pulse generator(Handwheel)

There are two types of handwheel, one is handwheel in handhold box; the other is handwheel in the operational panel.

Handwheel in Handhold Box: Press "Handwheel" key to enter handwheel mode. User can operate the axes selection & feeding override of handwheel(*1/*10/*100).

Handwheel is mainly used for "Tool", also posit tool.

Speed of handwheel pulse generator should be lower than 200r/min(100 pulses per cycle).

Parameters set for handwheel

P1 in Other parameter is set for position of handwheel.

In Speed parameter:

P23 is set for acceleration / deceleration time constant.

P24 is set for acceleration / deceleration time constant when simulate.

P25 is set max speed of X-axis with handwheel.

P26 is set max speed of Y-axis with handwheel.

P27 is set max speed of Z-axis with handwheel.

P28 is set max speed of A-axis with handwheel.

In Axis parameter:

P311 is set max speed of B-axis with handwheel.

Note:

1. SZGH-CNC1000MDc series support handwheel both in panel & in handhold box.

2. Handwheel is no effect in auto-coordinates diagram machining, it only works in mode of coordinates.

3. When system is configured with stepper system, feeding speed shouldn't be too fast.

Fig1.3 Handhold Box(Manual Pulse Generator)

1.5.4 Manual Reference Position Return

The CNC machine tool has a position used to determine the machine position. This position is called the reference position, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on or alarm/release emergency stop, the tool is moved to the reference position.

Manual reference position return is to move the tool to the reference position using switches and push-buttons located on the operator's panel.

In axis parameter, P38 is set for the grade of homing; there are 4 grades for homing. 0: Just hint; 1:No use; 8: Compulsion; 9: Super Compulsion;

Suggestion: In order to ensure processing accuracy, it needs homing after reboot.

There are two ways that return to reference position manually, one is that floating zero point, the other is that switch for homing, details as follows:

(1) Floating Zero Point Set

When user don't fix switches for Homing , user can use floating zero point as home, also reference position, also original point of machine coordinate system.

Parameters set: In Axis parameter,

P33: Bit parameter, D3:X ; D4:Y ; D5:Z ; D6:A ; D7:B ; 1 means float zero point;
P34: Distance between reference position & current position in X direction;
P35: Distance between reference position & current position in Y direction;
P36: Distance between reference position & current position in Z direction;
P37: Distance between reference position & current position in A direction;
P318 in Axis parameter:Distance between reference position & current position in B direction;

Steps of setting floating zero point(home):

1. Enabled function of float zero point of all axes(XZCAB): P23= 11111001;
2. Moving each axes to designated position in order to set floating point easily.
3. Set distance between reference & designated position: P34/P35/P36/P37/P318

Note: If current position is home of machine coordinate system, no needs to set offset of P34/P35/P36 /P37 /P318 for each axis.

(2) Switch For Homing Set

User can fixed switches for homing,some parameters & wiring needs to done.

A. Parameters set: **in Axis Parameter**,

P33: Bit parameter, D3:X ; D4:Y ; D5:Z ; D6:A ; D7:B ; 0 means switch for homing;
P38: Grade of Homing,0: Just hint; 1:No use; 8: Compulsion; 9: Super Compulsion;
P39: Mode of Homing, 0 & 2: Detect Z0 signal; 1 & the others, no detect ;
P40: Bit Parameter, Direction & Sequence of Homing;
P41: Bit parameter, Type of switch for homing ;
P42: Range of detecting Z0 signal in X-axis ;
P43: Range of detecting Z0 signal in Y-axis ;
P44: Range of detecting Z0 signal in Z-axis ;
P45: Range of detecting Z0 signal in A-axis ;
P46: Offset after homing in X-axis
P47: Offset after homing in Y-axis
P48: Offset after homing in Z-axis
P49: Offset after homing in A-axis

In Speed Parameter,

P34: X-axis homing speed

P36: Y-axis homing speed

P38: Z-axis homing speed

P40: A-axis homing speed

P35: Speed during detecting Z0 signal of X-axis

P37: Speed during detecting Z0 signal of Y-axis

P39: Speed during detecting Z0 signal of Z-axis

P41: Speed during detecting Z0 signal of A-axis

In Axis parameter, about all parameters for B axis

P314: Range of detecting Z0 signal in B-axis ;	P315: Offset after homing in B-axis
P312: B-axis homing speed	P313: Speed during detecting Z0 signal of B-axis

Note: Details about parameters set,please check part of Parameters List(see chapter 2)

B. Input points for homing in the table

Input Point	PIN	Function
X0	PIN3 of CN3 Plug	X axis homing
Y0	PIN2 of CN3 Plug	Y axis homing
Z0	PIN17 of CN3 Plug	Z axis homing
M34/A0	PIN4 of CN3 Plug	A axis homing
M24/B0	PIN3 of CN10 Plug	B axis homing

Note: 1.when system hasn't A axis & B axis, the input point is as function of M34/M24.

2. Valid level of all input points is 0V, also common port is 0V or NPN type .

3. Wiring for homing, please see Appendix I Wiring Diagram of CN3 Plug.

C. Operation of Return Reference Position

Press "Return" in Manual mode,system will hint "Input axis name:(X; Y; Z ; A; B; 0(ZXYAB))" , user can select one axis for homing alone, and also input "0" & Enter key , all axes go homing sequentially.

Note: 1. If user want to stop during homing, press "Emergency Stop" or "Reset" to stop.

2. After homing successfully, circle before coordinate will change to green, otherwise homing is failure.

The tool can be moved to the reference position also with program commands.

This operation is called automatic reference position return(See Chapter3.15_Book1).

Special Attention: Every time to power up the system must back to zero point to make sure the accuracy of machine processing. The system power off unusually or in an accident, it must return to reference point ,also homing, otherwise could cause trouble.

1.5.5 Setup Workpiece Coordinate System

Steps of setup workpiece coordinate system

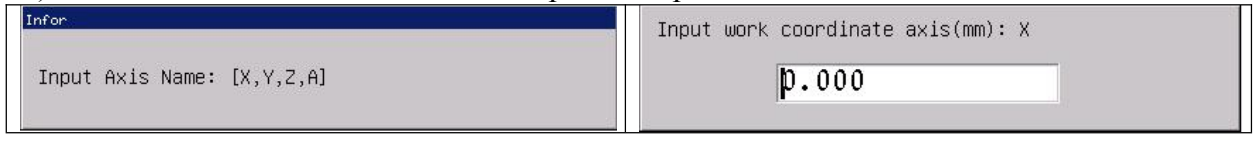
1) Press "F5_MDI" key to enter MDI mode on Manual Condition

2) Input "G54 ~ G59"or Press "F8 ChoCo" to select workpiece coordinate system

3) Move feeding axes that need to posit tool to specified position

4) Press "F7-SetCo" key , and select related axis and input current coordinate position

5) Press "Enter" to ensure and finish setup of workpiece coordinate.

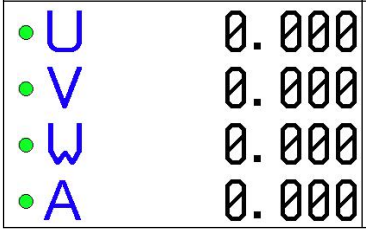
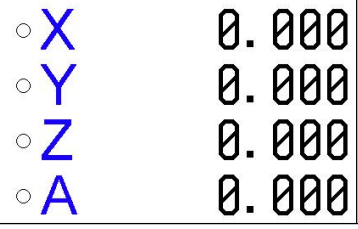
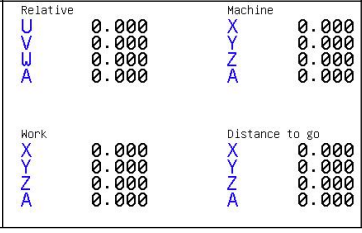
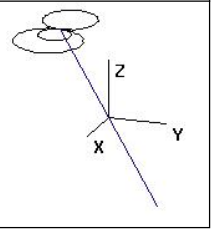


1.6 Auto Operation

Auto refers to processing the editing program of workpiece. This system can start at arbitrary point, and also can start at arbitrary line or with arbitrary tool. Starting arbitrary line or with arbitrary tool must use absolute coordinate to edit the program. Press "Auto" to enter Auto mode in Manual mode. User can't move coordinate manually in Auto mode.

Select running program: Press "Program" key to enter program interface, press "↑ ↓" to move cursor to a program which is going to run , press "C" key to select the program as processing program and switch to main screen automatically.(See Chapter 2. 4_Book1)

Switch display of coordinate: Press "PgUp"/ "PgDn" to switch the display which correspond to "Relative" "Absolute" "All". & "Graphic".

Relative Coordinate	Absolute Coordinate	All Coordinate	Graphic
			

1.6.1 Automatic Processing Mode

"Single/continuous": Press "Single" key to switch cycle.

"Continuous": The program continue to execute every program segment(program line) to end or the instruction of stop to stop.

"Single ": The program just execute one program line and end, wait another operation or press "Run" again to execute one next program line.

"Simulate": The program is speedy simulate, the axis of coordinate can't move.

Table of Status Display about Manual/Auto

Manual Contious	Manual Increment	Handwheel	Feed Hold
Man Con	Man Inc 1.000	Man PulsX *100	Man Con Keep
Auto Continous	Auto Single	Simulate	
AutoCon	AutoStepStop	Imi Con Stop	

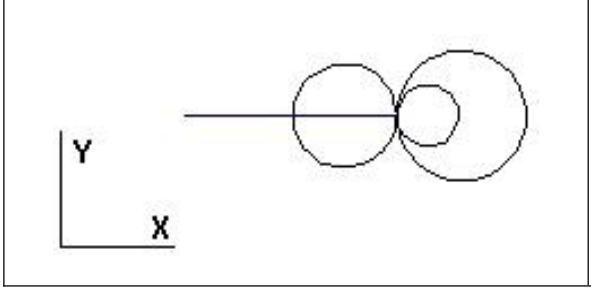
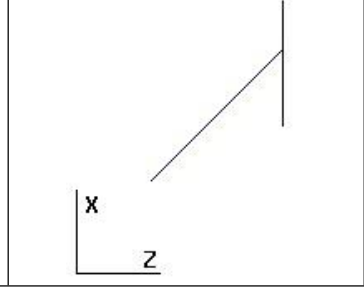
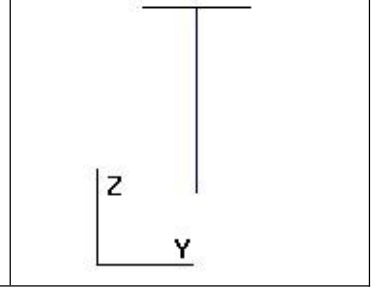
"Coordinate/Graphic": Press "Graphic" in Auto mode to switch cycle.

"Automatically coordinate": The axis of coordinate will display with value.

"Automatically Graphic": The axis of coordinate will display with a figure.

"Feed Keep", also feed hold, Press "Manual" twice on "AutoCon Run" condition or Press "Manual" on "AutoCon Stop" after running.

Press "X" "Y" "Z" on "Graphic" mode to shift 2D mode(XY,ZX,YZ) plane, press "0" back to 3D.

XY Plane	ZX Plane	YZ Plane
		

1.6.2 Processing at arbitrary program line or with arbitrary tool

1.6.2.1 Start from "nth" line(block)

At the condition of automatic processing, press "—" to pop-up a dialog box, import a number of line, press "Enter" to confirm, system will start program from this line,and display at processing program. No.1 line of program is 1th line , input number is nth line , as one block is one line.

Note: ***1. The line is the actual line in the program, not the "N" stand for the line.***

2. Firstly of all, system will move the starting point of "nth" block with speed which is set by P7 in Speed parameter, then run the program normally.

3. If user don't input line number, CNC system will jump to the line that program stop last time, to facilitate user's operations.

4. Press "Reset" to return back to the beginning of program after use "N" to search line number in coordinate screen.

1.6.2.2 Start from "N**" line

The system support that program can start from "N**" line(N** is 4 digit marked line) . At automatic process condition, press "N" to pop up a dialog box to import the marking line, press "Enter" to confirm. Press "Start" to run program at the "N**" line you import(mark).

Note: 1. "N**" line is not "nth" line/block, is the "N**" stand for the line.

2. Firstly of all, system will move the starting point of "nth" block with speed which is set by P7 in Speed parameter, then run the program normally.

1.6.3 Start Program

Start program must in the mode of "Auto" , press "Auto" key to enter mode of Auto, there are two methods to start program, as follows,

(1) Press "Start" key in the operational panel.

(2) Fix external switch to Run port(PIN18_CN3/PIN8_CN6/PIN9_CN11)

Note: PIN9_CN11 can be used as Run port, P33 in Other parameter is set for this function.

1.6.4 Halt Program

There are five methods to stop/pause program , as follows,

(1) Instructions of program: M00, M01, M02, M30, M20.

(2) Press "Single" to run a current block and stop.

(3) Interface switch turn to the middle or right side.

(4) Press "Reset" to stop all the actions of program.

(5) Fix external switch to Halt port(PIN6_CN3/PIN6_CN6/PIN10_CN11).

Note: PIN10_CN11 can be used as Halt port, P34 in Other parameter is set for this function.

1.6.5 Emergency Stop

Press "Emergency Stop" button when emergency accidents happening, the system will stop all the actions of machine tool and shows "Emergency stop" on screen.

User can fix external switch as Emergency Stop, Input point is PIN5 of CN11 plug.Type of emergency stop switch can be set by P27 in Other parameter.

After "Emergency Stop" during processing, which will affect difference between machine tool coordinate system & actual position of machine tool, in order to ensure coordinate system is same

to actual position of machine tool, it is best to do manual return reference position(homing) before processing.

1.6.6 Alarm

The screen hints alarm message when machine tool alarm, CNC system will stop processing. Only after clear alarm,and then CNC system can processing.

There are some fixed alarm,cannot be changed ,as following

Alarm Hinting Message	Input Point
X-axis driver has happened hardware limit	-L(PIN15_CN3 Plug) +L(PIN16 _CN3 Plug)
Y-axis driver has happened hardware limit	
Z-axis driver has happened hardware limit	
A-axis driver has happened hardware limit	
B-axis driver has happened hardware limit	
X-axis,Y-axis,Z-axis,A-axis,B-axis driver is alarming	ALM(PIN12_CN5 Plug)
Spindle driver is alarming	ALM1(PIN5_CN3 Plug)
Emergency Stop	Stop(PIN5_CN11 Plug)

There are also some user-defined alarm as following

Alarm Hinting Message	Auxiliary Relay in PLC
No.0 Alarm	M80
No.1 Alarm	M81
No.2 Alarm	M82
No.3 Alarm	M83
No.4 Alarm	M84
Protect Door Is Open	M85
No.6 Alarm	M86
No.7 Alarm	M87
Loss of Lubricate Oil	M88
No.9 Alarm	M89
No.10 Alarm	M90
No.11 Alarm	M91
+5V Under-voltage	M92
+24V Under-voltage	M93
No.14 Alarm	M94
No.15 Alarm	M95

Note: these alarm hinting message & input point can be edited as user's asks on PLC.

Input point for protect door: M12, PIN11 of CN10 Plug

In Other parameter:

P7: Detect switching signal of protective door, 0:no detect,1: yes

P8: Type of switch for protective door, 0: NO type, 1: NC type.

P17: Type of alarm for servo driver, 0: NO type, 1: NC type.

P18: Type of alarm for spindle, 0: NO type, 1: NC type.

P19: Type of alarm for machine tool, 0: NO type, 1: NC type.

P26: Type of switch for emergency stop in panel, 0: NO type, 1: NC type.

P27: Type of switch for emergency stop in CN11, 0: NO type, 1: NC type.

Note: Emergency STOP: Press "Emergency STOP" button when appearing emergent accident, the lathe will stop all actions and the screen of system shows "Emergency STOP". Wait for releasing the button.

1.6.7 Indicator Light Output

Output Signal	Output Point	Parameter Set
Program Running	M69(PIN21_CN10 Plug)	P28=1 In Other parameter
Program Halt	M65(PIN20_CN10 Plug)	
Alarm	M67(PIN8_CN10 Plug)	P29=1 In Other parameter

Note: more details about indicator light output, please check Chapter 4.3.1.7.

1.6.8 DNC function

Storage room of SZGH CNC system is 128Mbit, user can adopt RS232-DNC or USB-DNC function to run the processing program that is greater than the remainder storage. RS232 port & USB port are in the front of CNC1000MDc series controller.

1.6.8.1 RS232-DNC

1. Connect PC and CNC system well with the dedicated communication line , & set communication rate by P37 in Other parameter;
2. Use the dedicated communication software(SZGHCNCCS) on PC to set the related communication port and rate. Press “Transmit CNC”, select the program file to process linked, enter the status of sending program file.
3. To enter the interface of program file on CNC system, press "L" to enter the status of linked process, and program will display "RS232--DNC", press “Start” to running carry out linked process in the automatic status.
4. Turn “Interface switch” to middle or right to stop the running system in the process of linked process, press “E-Stop” or “Reset” to exit link of DNC.

Note: 1. Baud rate is related to operational environment when using serial port to send files.

2. The communication cable can’t more than 10 meters length.

3. Only the dedicated communication software of this system can send program in User’s computer. To set the sending speed of PC as the NC, defeat otherwise.

1.6.8.2 USB-DNC

USB-DNC is realized by U-disk, switch on U-disk and system, select program to execute in U-disk.

Press “B” to open U-disk in program interface, select corresponding program to press “C” to execute program, press “Auto” to get into automatic mode and press “Start”/ “Run” button to process the program.

Note 1. Don’t unplug U-disk in the process of USB-DNC, otherwise failure.

2. Back to the system program interface from U-disk interface after finish USB-DNC.

3. After selecting the program, it is best to press “P” to compile once to make sure the program is right before executing program of USB-DNC.

1.7 External Electrical Connection

Basic IOs of this CNC system is 56*32, some input points & output points have related functions.

1.7.1 Limitation

There are two ways to set limitation of machine tool , one is software limitation, another is fix external switch as limitation.

1.7.1.1 Software limitation

Software limitation is finished by setting working range of machine tool , also set related parameters in CNC system.

In Axis Parameter: P13: bit parameter, software-limitation of each axis is set alone.

Bit_P11	D7	D6	D5	D4	D3	D2	D1	D0
Axis		B	A	Z	Y	X		
Default set	0	0	0	0	0	0	0	1

1: it is invalid of software-limitation,0: valid of software-limitation.

P3: Max Range in X-Negative Direction
P4: Max Range in X-Positive Direction
P5: Max Range in Y-Negative Direction
P6: Max Range in Y-Positive Direction
P7: Max Range in Z-Negative Direction
P8: Max Range in Z-Positive Direction
P9: Max Range in A-Negative Direction
P10: Max Range in A-Positive Direction

Warning: when shift Metric/Inch, unit is changed(mm/inch), data for limitation don't change.

1.7.1.2 External Switch for limitation

Input Point of Limitation

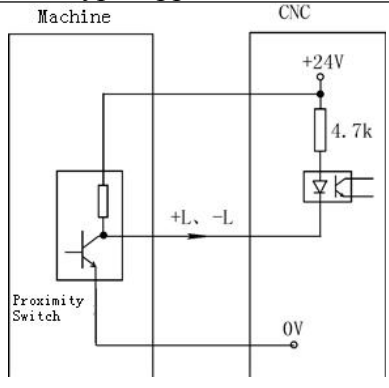
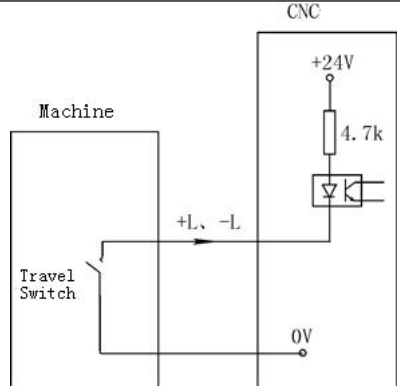
Mark	Port	Explanation
-L	PIN15_CN3 Plug	Limitation in negative direction
+L	PIN16 _CN3 Plug	Limitation in positive direction

Type of Limitation Switch

In Axis parameter:

P21, Type of switch in Positive direction [0: NO type, 1: NC Type]
P22, Type of switch in Negative direction [0: NO type, 1: NC Type]

Wiring of Limitation

NPN Type Approximate Switch	General Switch
	

1.7.1.3 Suggestion Usage

Fix hardware limitation switch within the range of sets by software-limitation, P11=00000001; even if hardware switches don't work, software-limit also will work, double safe for limitation,

Note: 1. When limitation switch is NO type(normal open type) switch,use parallel connection; when limitation switch is NC type(normal close type) switch, use series connection.

2. CNC system could define X0&Z0 as limitation of X/Z axis. X0 signal as limitation and home of X axis, controlled by one switch. Z0 signal as the limitation and home of Z axis, also controlled by one switch.It needs to restore our special PLC ladder into CNC system.

In Axis parameter:

P27, Type of Switch in positive direction, also for +L [0: NO type, 1: NC Type]

P28, Type of Switch in negative direction, also for -L [0: NO type, 1: NC Type]

1.7.1.4 External Switch for Power ON/OFF

It needs to use one contactor(KM1) & two switches for turn on & turn off power, one is NO type, which is for turn on power, another is NC type, which is for turn off power.

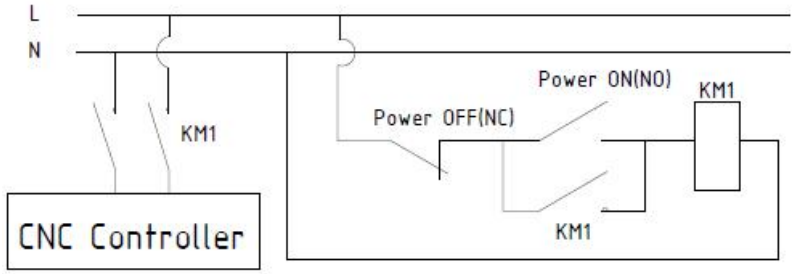


Fig1.7.1 Wiring Diagram for Power ON/OFF

1.8 Diagnosis

Press “Diagnosis” key to enter the diagnosis interface.

Press “F2” & “PgDn/PgUp”or “↓ ↑” to check the status of inputs and outputs.

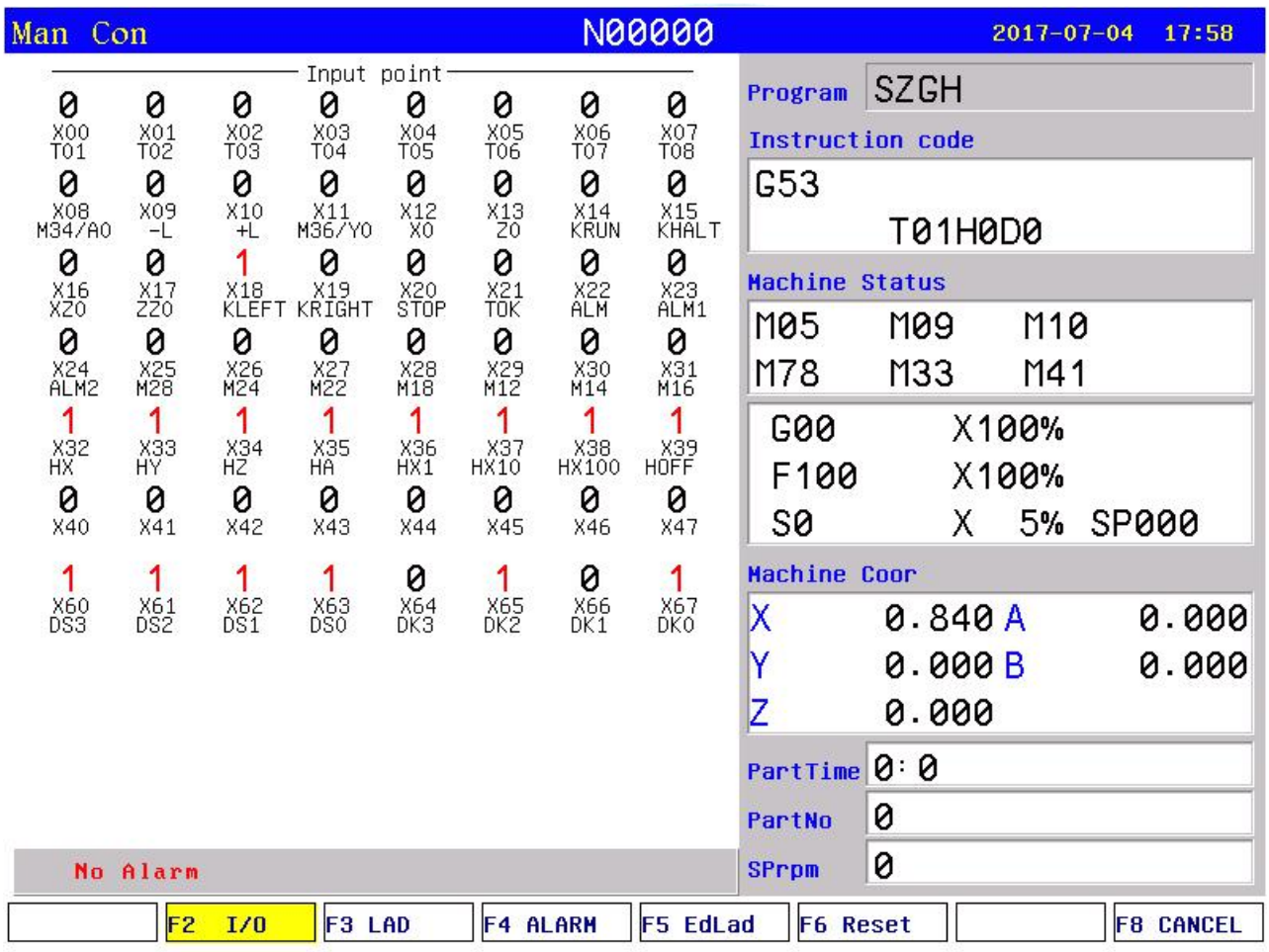


Fig1.8.1 System Diagnosis Interface(Input signal)

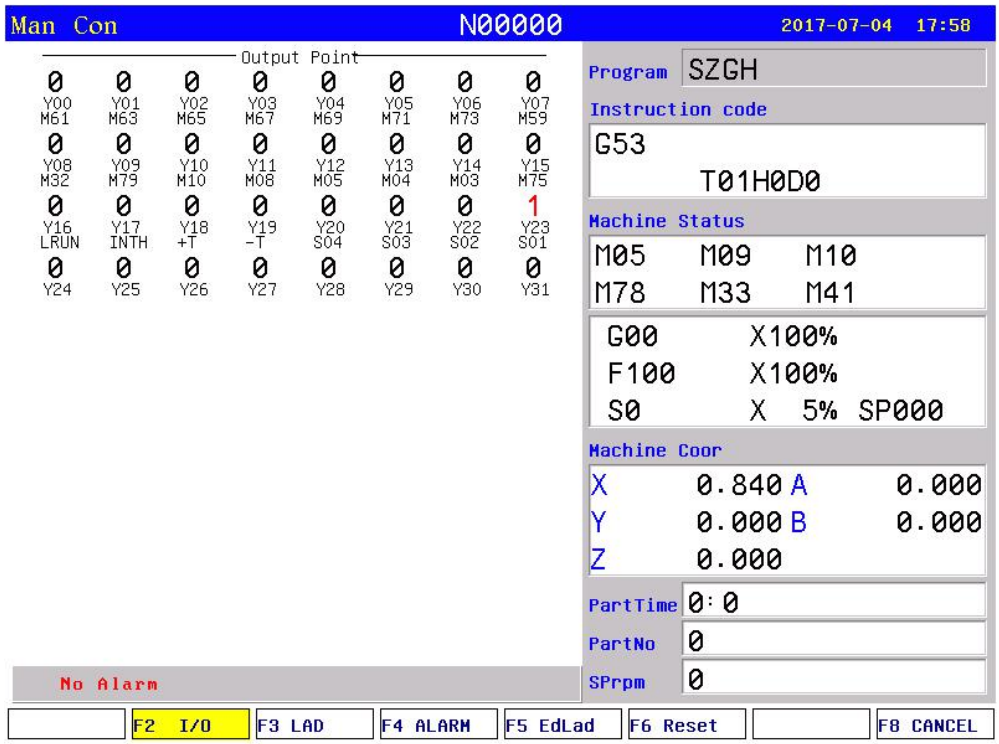


Fig1.8.2 System Diagnosis Interface(output signal)

In diagnosis interface of I/O , “0” means invalid status, “1” means valid status.

Press “F3” key diagnosis screen to enter interface of check condition of PLC.

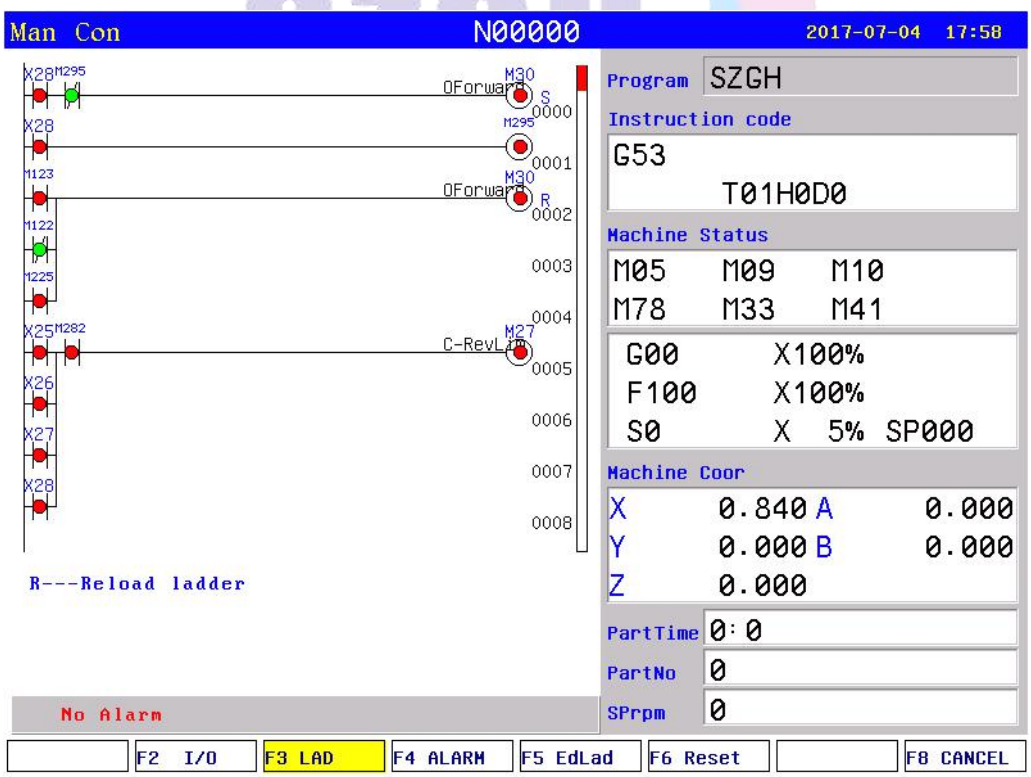


Fig1.8.3 Condition of Inner Register & IOs

Press “PgDn”, “PgUp” , “Up arrow” , “Down Arrow” to check condition of inner registers & outputs & inputs.

“Green” means the register is valid, “Red” means the register is invalid.

Press “F5” key on diagnosis screen to enter interface of edit ladder of PLC.

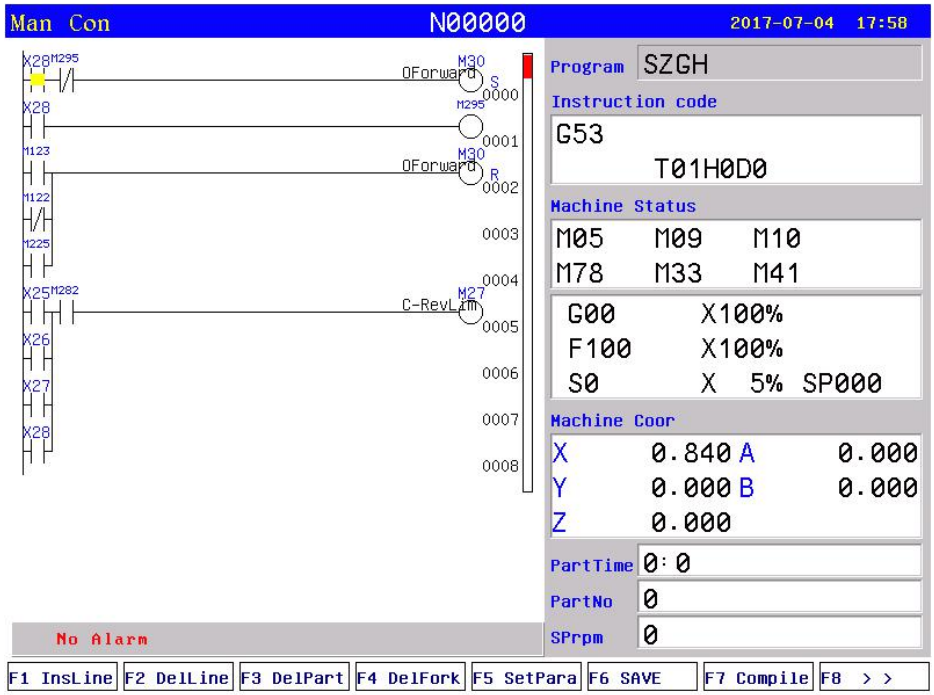


Fig1.8.4 Editing Screen of Inner Ladder

Press “S” key on these interfaces to activate search function. After finish ladder & save, it will work after reboot.

Press “R” key on condition screen of PLC, PLC will work immediately & no needs to reboot.

Note: when P1 in Password parameter set to Disable, and then user can check & edit inner ladder.

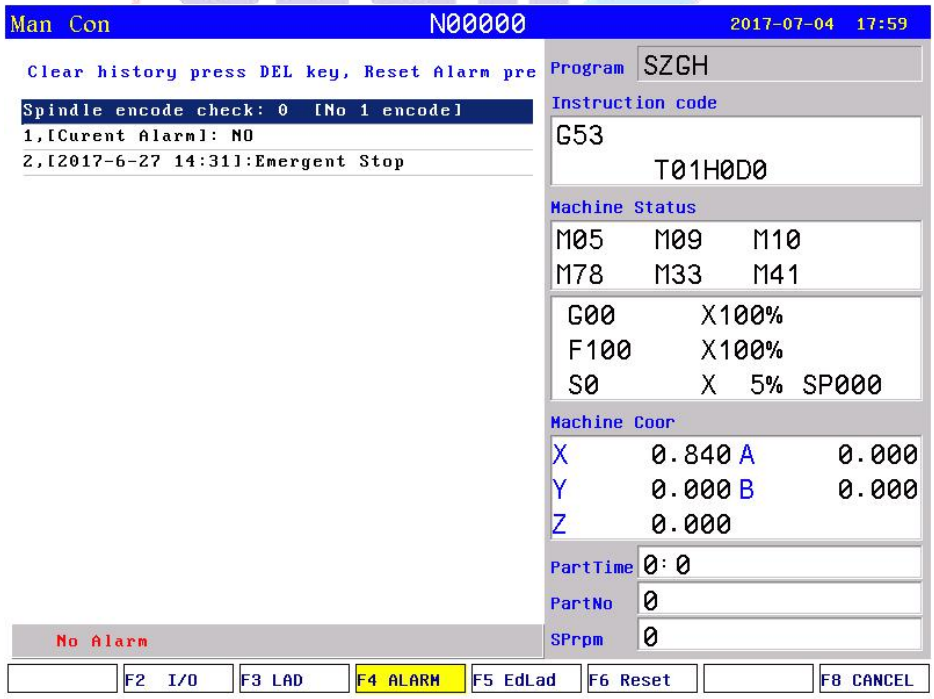


Fig1.8.5 System Diagnosis Interface(Alarm messages)

Spindle encode check:0 ,shows current resolution of SP_encoder;

1,[Current Alarm]:NO, shows current alarm condition of machine tool

Following shows history alarm messages.[Press Del key to clear alarm messages]

1.9 Programming Operation

Press “Program” to enter into status of Program as following.

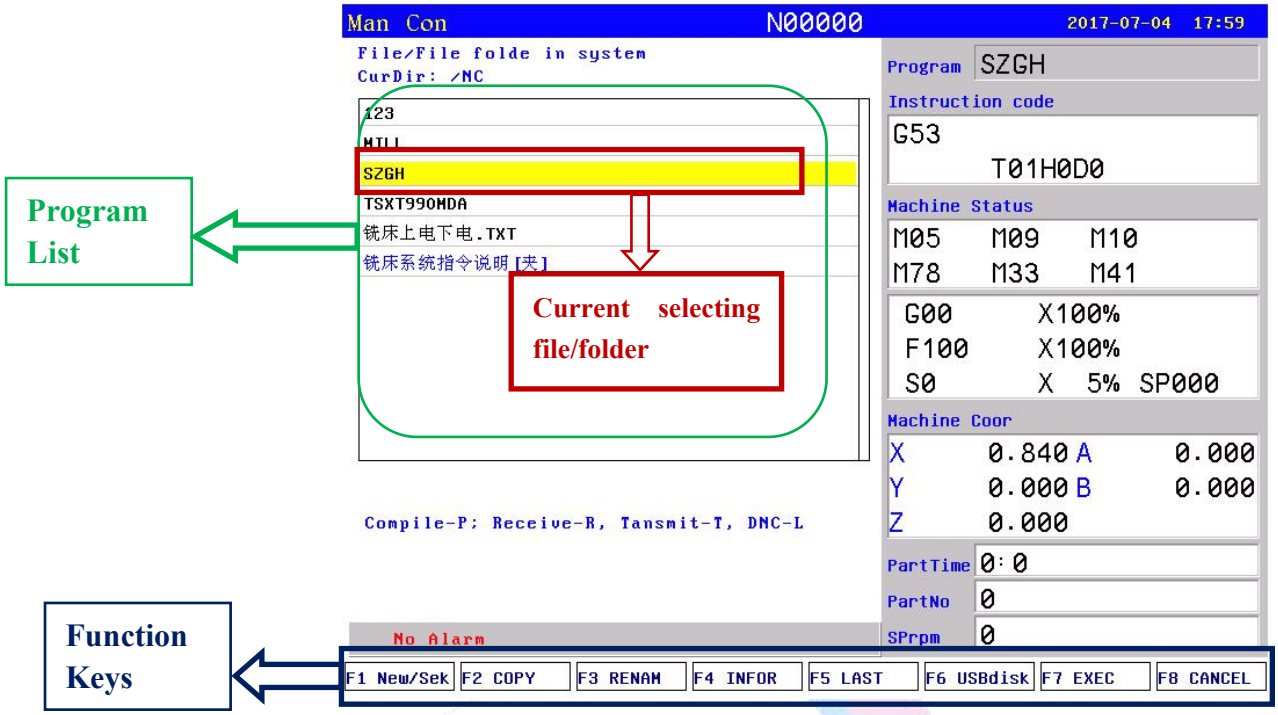


Fig1.9.1 Interface of Program

Management of program adopt mode of file/folder management, storage room of SZGH CNC system is 128Mb, there is no limitation about quantity of programs.

At program list,press “PgDn/PgUp”or “↓,↑” to select program/file.and then press “Enter” to enter current program.

Name	Function
New/Sek	Press “F1” key to New/Search a program
COPY	Press “F2”key to Copy selecting program(System ↔USBdisk)
RENAM	Press “F3” key to Rename current selecting program
INFOR	Press “F4” key to hints size of program& remain space of system
LAST	Press “F5” key to return to last level
USBdisk	Press “F6” key to open USB-disk
EXEC	Press “F7” key to execute current program
CANCEL	Press “F8” key to cancel or return
Compile-P	Press “P” key to compile current program
Receive-R	Press “R”key” to receive file from PC with RS232-DNC
Transmit-T	Send file to PC with RS232-DNC
DNC-L	Open function of RS232-DNC between PC & CNC

1.9.1 Editing

Press “F1”key and pop-up a dialog box to input the name of program, if the name is existing, the existing program is called up; If the name isn’t existing, the system will build a new program.

The name of program could be number, letter or mix, the length is 100 bits.

Build a new program or select a program and press “Enter” to entering the editing interface.Press “F7” button to shift to function interface.

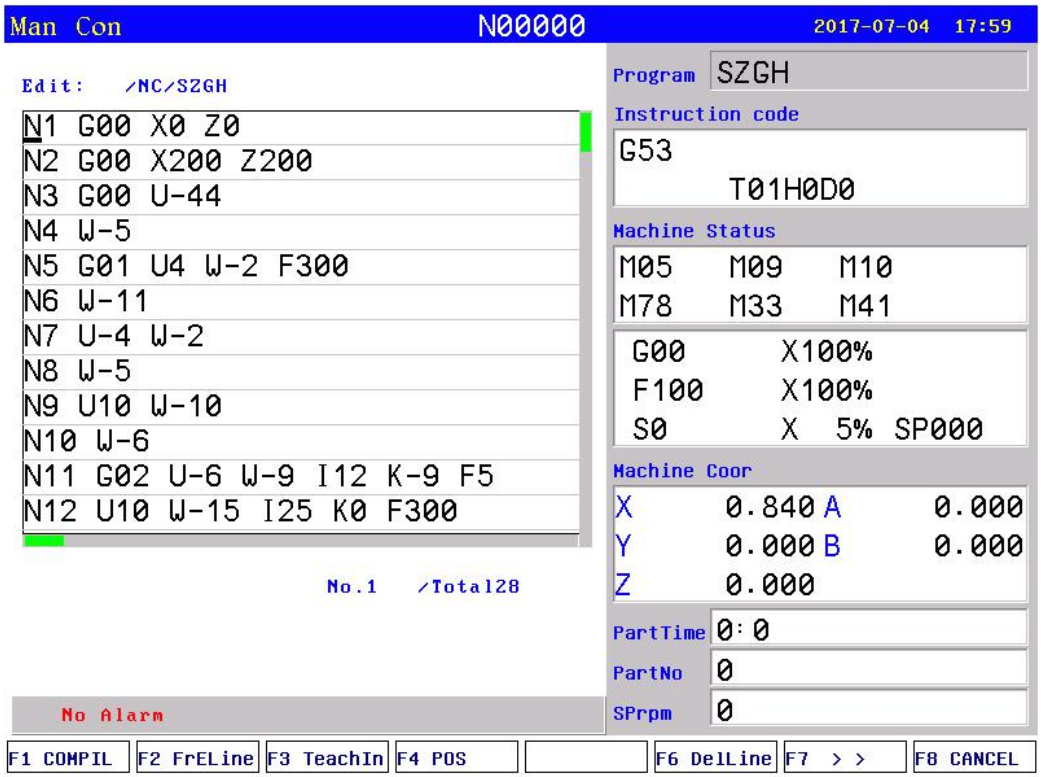


Fig1.9.2 (1)Editing interface of Program

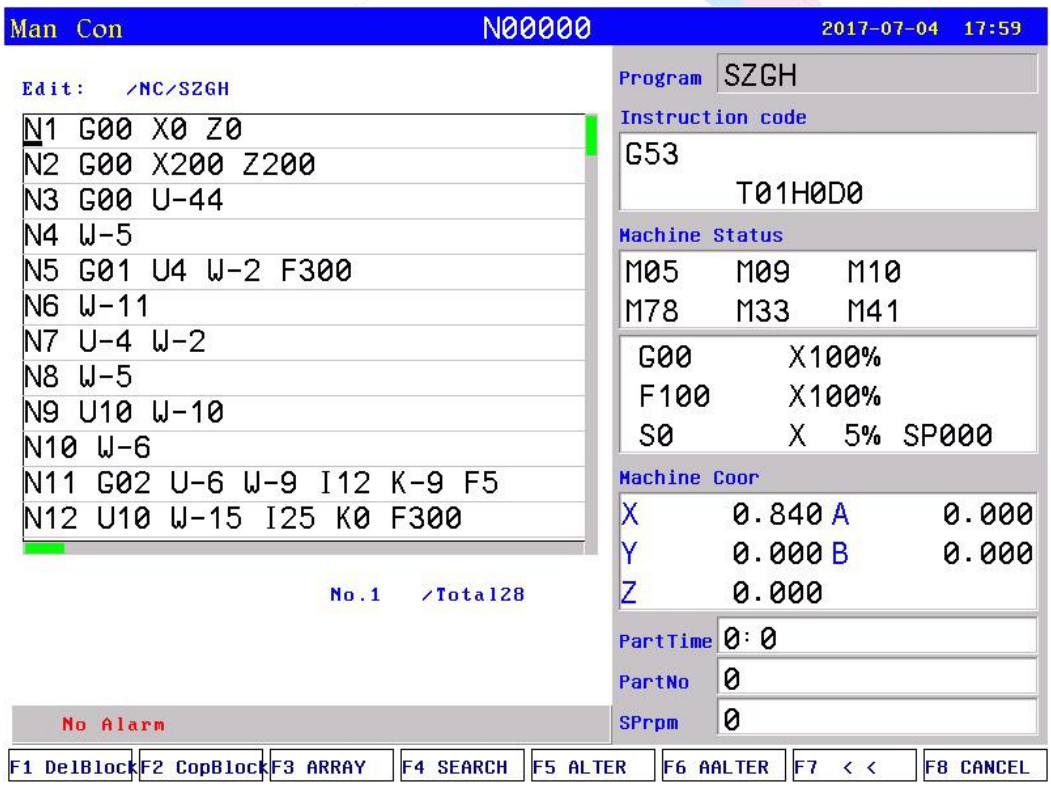


Fig1.9.3 (2)Editing interface of Program

Note:The name of all files don't allow same & blank.

The screen prompt the editing program name at the top left corner in the editing status; The left is the content, the right is the information for status, the operation in the editing status as follows:

(1) Location of current cursor:

Press “↑ ↓←→” to move the cursor to any position of program content

Press "Pgup" to the last page.

Press "Pgdn" to the next page.

(2) Character Modification: Delete the character at the position of the cursor, then enter the new character.

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

(4) Character Deletion: Press "Del" directly at the cursor position

(5) Inset a line: Press "Enter" directly, inset a line in front of the current line if the cursor is at the first line, otherwise insert a line after the current line.

(6) "Rapid" key Overlay operation:

First Function Keys	
	Function
F1	Compile current program
F2	Cursor jump to first line or last line
F3	Teaching function, enter status of Handwheel;Press X/Z/Y/C/A key to shift selecting axis, and press “Rapid”+ “Q” to read & set value of current coordinate system.
F4	Cursor jump to the specified line
F5	Null
F6	Delete current block.
F7	Shift first function keys & second function keys
F8	Cancel
Second Function Keys	
F1	Delete specified blocks from current line to input line
F2	Copy specified blocks from input begin line to input last line
F3	Array all blocks of current program
F4	Search specified characters from cursor line to end line
F5	Replace 1st specified characters from cursor line to end line
F6	Replace all specified characters from cursor line to end line
F7	Shift first function keys & second function keys
F8	Cancel

1.9.2 Copy

Press “↑ ↓” in main interface of Program, to select the program which need to copy and press “F2” to pop-up a dialog box to import a new name of program, to copy which is the same content but different name , in order to modify, rename and spare.

1.9.3 Delete

Press “↑ ↓” in main interface of Program, to select program which need to delete and press “Del” key to delete the program.

Note: The operation of delete need to be careful, it can’t be recovery once deleted.

1.9.4 Rename

Press “↑ ↓” in main interface of Program, to select program which need to rename and press “F3” to pop-up a dialog box to import a new name.

1.9.5 Information

Press “↑ ↓” in main interface of Program, to select program which need to check and press

"4" to pop-up a dialog box to check the size of current program and remain space of the system.

1.9.6 Compile

Press " ↑ ↓ " in main interface of Program, to select program and press "P",or Press "F1" key on editing interface of program, the system will check the format and grammar of program. Prompting when finding mistake automatically.

Note: Before run program, it should compile successfully.

1.9.7 Folder management

User can build a folder in this system, Press "F1" in main interface of Program to import a name of folder ,add "." , and press "Enter"to build a folder and it will prompt a "[夹]" after the name.

Note: the name of folder must be different to name of other file/folder,otherwise failure.

Move the cursor to the folder and press "Enter" to open folder , user also can build a new file or folder in this folder.

Press "A" go to the last folder.

Move the cursor to the folder and press "Del" to delete the folder.

1.9.8 Execute Program

Press " ↑ ↓ " in main interface of Program to select a program and press "C" to select the processing program and switch into main interface of CNC system.

1.9.9 Communication

The system could deliver files with RS232 serial protocol .

There are two communication port for RS232 at front & rear of panel.

Remark	PIN	Function
RXD	PIN2 of Front DB9 Port	Receive Date
TXD	PIN3 of Front DB9 Port	Send Date
0V	PIN5 of Front DB9 Port	Ground

Delivery (Transmit)

Deliver the selected program in this system to another system or to PC for save. Press " ↑ ↓ " in main interface of Program to select the program and press "T" to deliver, press "Reset" to interrupt delivery.

Reception

Receive the selected program in another system or PC (Must be text file form). Press "R" to import a name of received program into the dialog box in main interface of Program , press "Reset" to interrupt reception.

Note: 1. Using the exclusive communication software to deliver program in User's PC.

2. The rate of deliver of both PC&CNC must be the same,otherwise failure easily.P37 in Other parameter is set the rate of CNC system.

3. The length of RS232 can't over 10 meters.

4. The number of serial port must be the same as the system setting.

5. Editing program of PC must be text file form.

1.9.10 U-disk management

To exchange files of parameter or programs with other system or PC by U-disk. It also can upgrade or back-up the software or parameter in system.

Note: The name of folder can't have blank symbols.

Suggestion: Please prepare special USB-disk for manage files for CNC system.

Press "F6"-USB-disk to enter the U-disk in main interface after U-disk is connected to USB port in the front of panel. Press "F6"-SYSTM again to back interface of system.

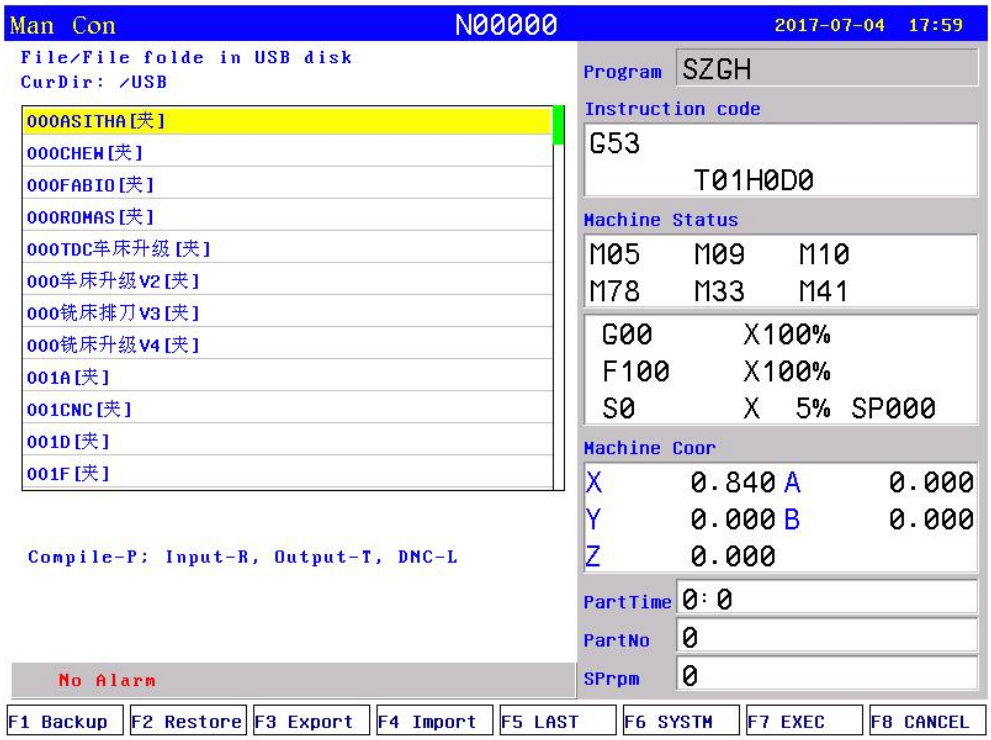


Fig1.9.4 Main Interface of USB-disk

1.9.10.1 Function Keys of USB-disk

Name	Function
F1-Backup	Press “F1” key to backup files of system to current directory of U-disk
F2-Restore	Press “F2”key to restore files at current directory of U-disk into system
F3-Export	Press “F3” key to export/copy file of system to U-disk
F4-Import	Press “F4” key to import/copy current file of U-disk to system
F5-LAST	Press “F5” key to return to last level
F6-SYSTM	Press “F6” key to return back to system,also exit USB-disk
F7-EXEC	Press “F7” key to execute current program at U-disk
F8-CANCEL	Press “F8” key to cancel or return
Compile-P	Press “P” key to compile current program
Input-R	Press “R” key to copy all files of U-disk to system
Output-T	Press “T” key to copy all files of system to U-disk
DNC-L	Open function of RS232-DNC between PC & CNC

1.9.10.2 Management of Processing Program

Copy the files or folder of U-disk into system

After connecting U-disk, press “F6” key to enter the U-disk directory in main interface of Program. Press “ ↑ ↓ ” to move cursor to select file or folder to copy and press “F4” key to pop-up a dialog box to import name, press “Enter” to confirm. If there is the same name of program in the system, it will pop-up a dialog box to ask if cover the file or folder or not.

Press “R” to copy all the program in U-disk into system.

Copy the files or folder of system into U-disk

Press “ ↑ ↓ ” to move cursor to select file or folder that needs to copy to U-disk, and then press “F6” key, press “F3” key to pop-up a dialog box to import name in U-disk interface and press “Enter” to confirm. If there is the same name of program in the system, it will pop-up a dialog box to ask if cover the file/folder or not.

Press “T” key to copy all the program in system to USB.

Note: 1. It must return to program directory of system, also exit U-disk by press “F6” key before unplugging U-disk, otherwise the date which is copied just now will be lost.

2. The name of folder can’t have blank symbol when using U-disk.

1.9.10.3 Management of Parameters & Software

User can use U-disk to deliver parameters files , system software , for upgrade and renew, back-up files and parameters of CNC system

A) Backup inner files & PLC files of system to U-disk

Note: Prepare a special empty U-disk for manage parameter files & PLC files of system better as Parameter files is lots of about several dozens, Or setup a folder in U-disk on your computer firstly, open the folder of U-disk on system before backup parameter files & PLC files into the folder of U-disk.

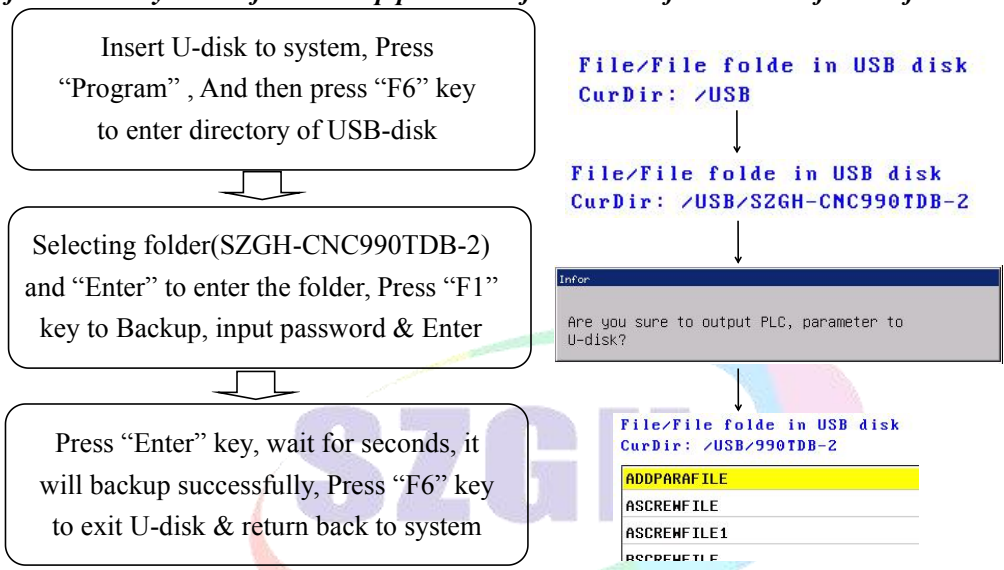


Fig1.9.5 Steps of Backup PLC & Parameter to U-disk

B) Restore parameters & PLC files into system with U-disk(Upgrade)

Note: Please put parameters & PLC for upgrade to a folder, which is better to avoid restore wrong files in U-disk into CNC system, and result to damage inner files & system.

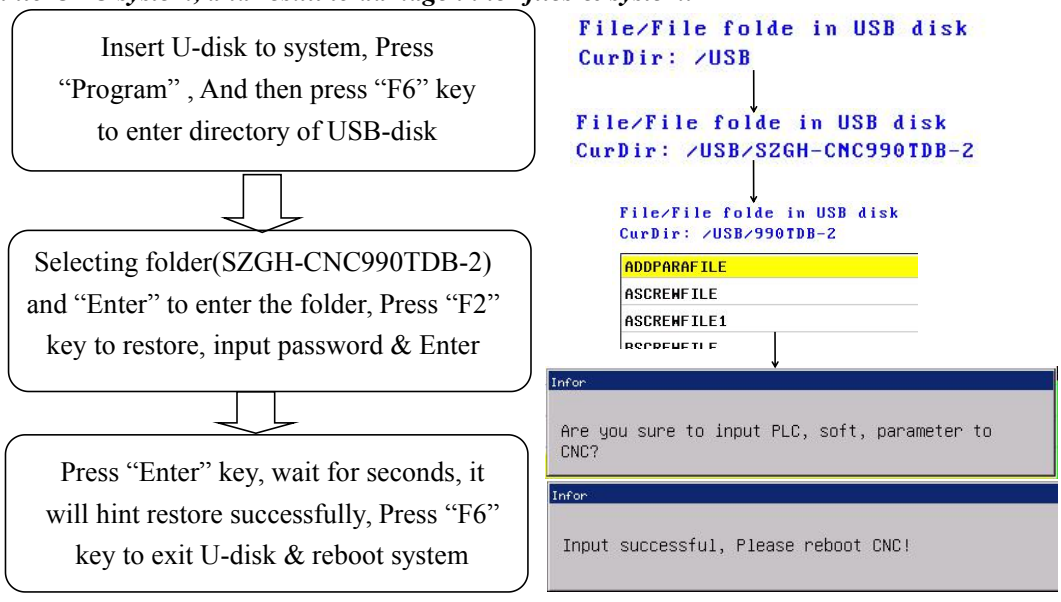


Fig1.9.6 Steps of Restore PLC & Parameter to U-disk

Chapter 2 Parameter List

At any status conditions, press “Parameter” to enter interface of parameter.

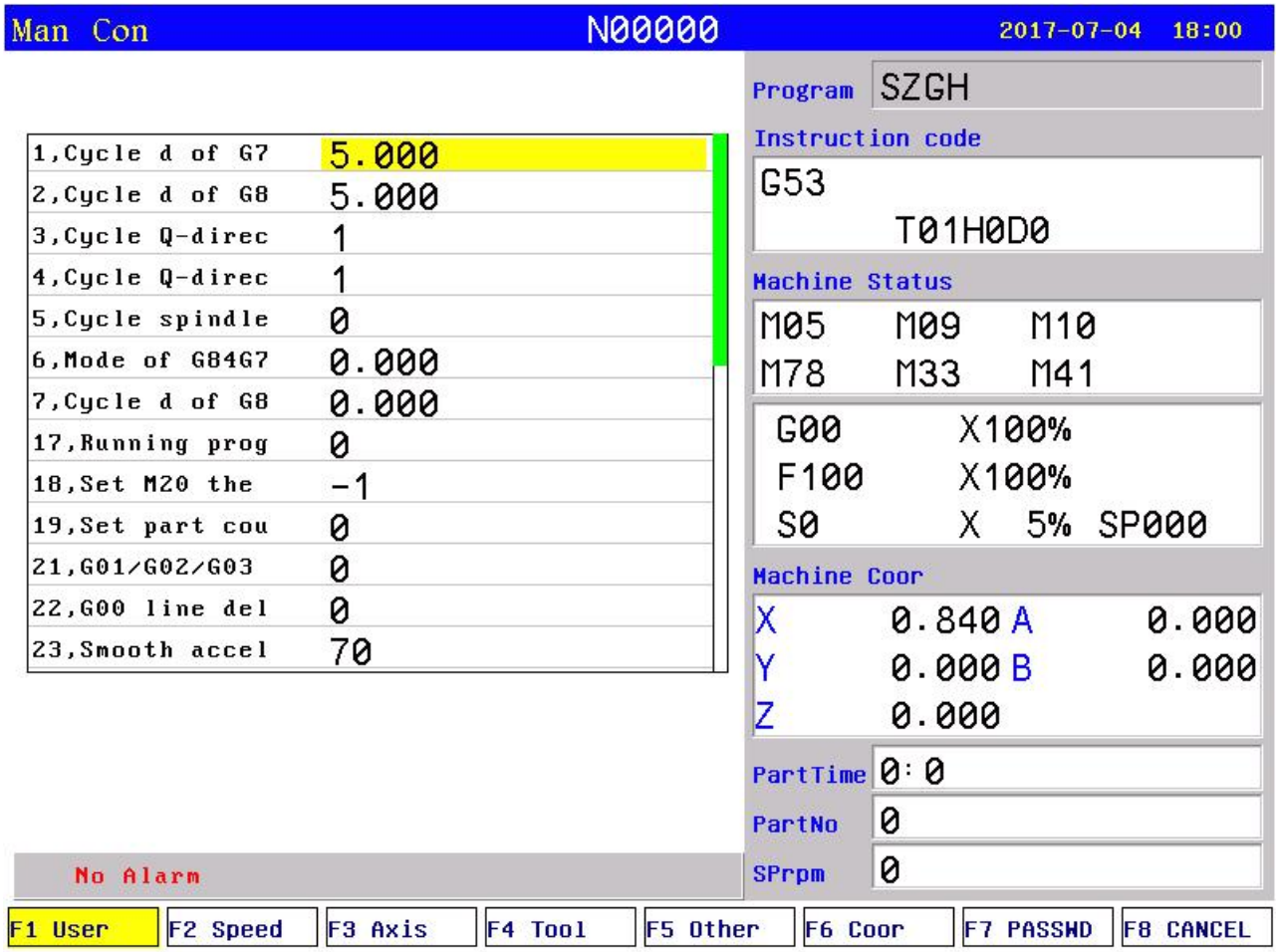


Fig2.1 Parameter List

Remark	Function
F1-User	Press “F1” key to enter User Parameter set
F2-Speed	Press “F2” key to enter Speed Parameter set
F3-axis	Press “F3” key to enter Axis Parameter set
F4-Tool	Press “F4” key to enter Tool Parameter set
F5-Other	Press “F5” key to enter Other Parameter set
F6-Coor	Press “F6” key to enter Coordinate Parameter set
F7-PASSWD	Press “F7” key to enter Password Parameter set
F8-CANCEL	Press “F8” key to exit parameter list

After enter Parameter interface & select kind of parameter list, press “ Down arrow ”, “Up arrow”, “PgDn”, “PgUp” to select one parameter that need to alter, Press “Enter” enter key , pop-up dialog box,after alter well, press “Enter” for ensure parameter is set well.

Note: 1.Yellow cursor means current Parameter kind & Parameter Number.

2.The version of Parameter List is V6.56.

2.1 User Parameter

P	Parameter	Ex-Value
1	(d)Escaping Amount of G73 (mm)	5.000
2	(d)Escaping Amount of G83 (mm)	5.000
3	Direction of shift amount Q in G76 [G17](1:+X,2:-X,3:+Y,4:-Y)	1
4	Direction of shift amount Q in G87 [G17](1:+X,2:-X,3:+Y,4:-Y)	1
5	Stopping Angle when Spindle orientation at G76/G87 (0.1degree)	0
6	Mode of clearance in G74/G84(0:Return with d; 8: Return to R)	0
7	(d)Escaping Amount of G74/G84 (mm)	0.00
17	Interlock between Running Program & SP-Rotating [1:Yes, 0:No]	1
18	Times of auto-cycle running(M20)[<0: Endless Loop]	-1
21	Dwell between G01/G02/G03 blocks(ms)[>100]	0
22	Dwell between G00 blocks (ms)[>100]	0
23	Acceleration/Deceleration Constant of Handwheel [50-100]	70
33	Type of Detect SP_Speed Reached(0:M69 Relay, 8:SP_Encoder)	1
34	Allow error of SP_Speed that detected by SP_Encoder(RPM)	1
200	Waiting time to screen saver [>=2minutes]	5
203	Inspect over-cut when tool radius offset (6326274:Yes,34818:NO)	6326274
210	Type of Graphic display area(8:manual,0:Automatic)	1
211	Display X-axis Negative area (1:Yes, 0: No)	1
212	Display X-axis Positive area (1:Yes, 0: No)	1
213	Display Y-axis Negative area (1:Yes, 0: No)	1
214	Display Y-axis Positive area (1:Yes, 0: No)	1
215	Display Z-axis Negative area (1:Yes, 0: No)	1
216	Display Z-axis Positive area (1:Yes, 0: No)	1
230	Running program through input point (+4+8+16+32+64+128: X26-X31)	0
231	Mode of "Delete" key [0:backward deletion,1:Forward delete]	0
232	Detect SP_Zero position before tapping [18:Yes,0:No]	1
233	G06 Circle teaching function[0:No, 1:Yes]	0
234	Activate Program Back Function with Handwheel[+8:Yes, Other: No]	9

Explanation about User Parameter(processing parameter)

1,(d)Escaping Amount of G73 (mm)

It sets every escaping amount in G73 code, unit:mm.

2,(d)Escaping Amount of G87 (mm)

It sets every escaping amount in G87 code, unit:mm.

3,Direction of offset Q in G76 [G17](1:+X,2:-X,3:+Y,4:-Y)

It sets the direction of offset Q in G76 code in XY plane(G17). 1:+X,2:-X,3:+Y,4:-Y.

4,Direction of offset Q in G87 [G17](1:+X,2:-X,3:+Y,4:-Y)

It sets the direction of offset Q in G87 code in XY plane(G17). 1:+X,2:-X,3:+Y,4:-Y.

5,Stopping Angle when Spindle orientation at G76/G87 (0.1degree)

It sets stopping angle when spindle do orientation at G76/G87 code. unit:0.1degree.

6,Mode of clearance in G74/G84(0:Return with d; 8: Return to R)

It sets mode of clearance in G74/G84;0:High-speed,return with d; 8: Normal, return to point R.

7,(d)Escaping Amount of G74/G84 (mm)

It sets every escaping amount in G74/G84 code, unit:mm.

17,Interlock between Running Program & SP-Rotating [1 mean Yes,0 mean No]

It is for interlock between run program and run spindle,when set as 1 means that running program with running spindle;when set as 0 means that running program without check spindle running.

18,Times of auto-cycle running(M20)

It is for times of run M20 code in the program,negative number mean run countless times.

21,Dwell between G01/G02/G03 blocks(ms)[>100]

It is for set delay time between G01/G02/G03,it is for solve the over-cutting in the corner.

22,Dwell between G00 blocks (ms)[>100]

It is for set delay time after run G00 ,it is effective that more than 100ms.

23,Acceleration/Deceleration Constant of Handwheel [50-100]

It is for set the constant of handwheel smoothly acc/dec-eleration. the smaller it is,the faster the acc/dec-eleration is,but much vibration.

33 Type of Detect SP_Speed Reached(0:M69 Relay, 8:SP_Encoder)

It sets the type of detect SP_Speed reached. 0 means that detect if M69(inner register of plc ladder) input is valid; 1 means that system detect spindle encoder for ensure spindle speed is reached.

34 Allow error of SP_Speed that detected by SP_Encoder(RPM)

It sets the allowable error of coding speed(S) & encoder feedback speed . Unit: rpm.

200,Waiting time to screen saver [>=2minutes]

It is the time that enter protection screen when system stay in main screen and without dialog,don’t enter screen protection if less than 2 minutes,press any keys to return back.

201,Delay time before detect zero pulse when threading(ms)[>100]

It is for set delay time before check Z pulse when process screw.

203,Using Pause key in Panel (23103490:Yes,6326274:No)

It is set for if using Pause key in operational panel, when set to 23103490, which means using Pause key; when set to 6326274, shield the Pause key.

210,Type of Graphic display area(8:manual,other:Automatic)

It set the type of graphic display area, when set to 8, the graphic display area is set

manually,& related parameter; when set to others, CNC system will adjust graphic display area automatically.

211,Display X-axis Negative area (1:Yes, 0: No)/212,Display X-axis Positive area (1:Yes, 0: No)
213,Display Y-axis Negative area (1:Yes, 0: No)/214,Display Y-axis Positive area (1:Yes, 0: No)
215,Display Z-axis Negative area (1:Yes, 0: No)/216,Display Z-axis Positive area (1:Yes, 0: No)

P210-P216 are set for if that CNC system display related area when type of graphic display area is manual.

230,Running program through input point (+4+8+16+32+64+128: X26-X31)

CNC system support run processing program by input points , related input points is X26-X31, Example: When P230=+4+8=12, inputs of X26 or X27 is valid, CNC system will running program of “X26”/“HIDEFILEX26” or “X27”/ “HIDEFILEX27”.

231,Mode of "Delete" key [0:backward deletion,1:Forward delete]

It sets the mode of “Del” , delete key, when set to 0, press “Del” key, system will delete backward word , when set to 1, Press “Del” key, system will delete forward word.

232,Detect SP_Zero position before tapping [18:Yes,0:No]

It is set for that if system needs to detect zero position of spindle encoder when tapping.

233,G06 Circle teaching function[0:No, 1:Yes]

It sets if system use G06 circle teaching function, 0 means no use; 1 means yes, use this function.

234,Activate Program Back Function with Handwheel[+8:Yes, Other: No]

It sets if system activate program return back function that back to front processing blocks with handwheel on Auto Handwheel condition.

Note:Press Auto & Handwheel keys enter this processing condition. Press Handwheel key again exit this processing condition.

2.2 Speed parameter

P	Parameter	EX-Value
1	G00 Speed of X-axis (mm/min)	4000.000
2	G00 Speed of Y-axis (mm/min)	4000.000
3	G00 Speed of Z-axis (mm/min)	4000.000
4	G00 Speed of A-axis (mm/min)	4000.000
5	Manual Max Feeding Speed(mm/min)	10000.000
6	Auto Max Feeding Speed(mm/min)	15000.000
7	Default Speed of G01/G02/G03 (mm/min)	2000.000
8	Running Speed at Simulation Mode (mm/min)	10000.000
9	Manual Feeding Speed(mm/min)	1000.00
10	Manual Spindle Speed (rpm)	0.000
13	Limit G01/G02/G03 Speed of each axis [1:Yes, 0:No]	0
14	Max Speed of X_G1/G2/G3 (mm/min)	2000
15	Max Speed of Y_G1/G2/G3 (mm/min)	2000
16	Max Speed of Z_G1/G2/G3 (mm/min)	2000
17	Max Speed of A_G1/G2/G3 (mm/min)	2000
18	X_Acceleration/Deceleration Constant [1～99999]	50000
19	Y_Acceleration/Deceleration Constant [1～99999]	50000
20	Z_Acceleration/Deceleration Constant [1～99999]	50000
21	A_Acceleration/Deceleration Constant [1～99999]	50000
22	Acceleration/Deceleration Constant When Auto Run	2
23	Handwheel_Acce/Dece-leration Constant [500--32000]	600
24	Handwheel_Acce/Dece-leration Constant when dry [>500]	600
25	G00 Speed when dry (mm/min) [>10]	1500
26	X_Max Speed with Handwheel (mm/min)	2000
27	Y_Max Speed with Handwheel (mm/min)	2000
28	Z_Max Speed with Handwheel (mm/min)	2000
29	A_Max Speed with Handwheel (mm/min)	2000
30	Type of Acceleration/Deceleration [0 mean line,8 mean curve]	0
31	Initial Acce/Dece-leration Constand when curve [>=10]	100
32	Quadratic Acce/Dece-leration Constant when curve [>=10]	500
33	Max Acce/Dece-leration Constant when curve [>=500]	1
34	X_Homing Speed (mm/min)	3000.000
35	X_Speed for detect Z0 signal (mm/min)	250.000
36	Y_Homing Speed (mm/min)	3000.000
37	Y_Speed for detect Z0 signal (mm/min)	250.000
38	Z_Homing Speed (mm/min)	3000.000
39	Z_Speed for detect Z0 signal (mm/min)	250.000
40	A_Homing Speed (mm/min)	3000.000
41	A_Speed for detect Z0 signal (mm/min)	250.000
42	Max Speed of 1st Spindle (rpm)	3000
43	Max Speed of Spindle at 2nd gear (rpm)	3000
44	Max Speed of Spindle at 3rd gear (rpm)	3000
45	Max Speed of Spindle at 4th gear (rpm)	3000
46	Max Speed of 2nd Spindle (rpm)	3000
47	Compensation Mode of Arc Backlash (0: A, 8: B)	8
48	Compensation Speed in B Mode (mm/min)	3000
48-1	Beginning Compensation Speed in B Mode (mm/min)	500
48-2	Acce/Dece-leration Constant in B mode (mm/min/s)	60000
49	Activate Speed Processing Function[1:Yes, 0:No]	1
50	Handwheel_Stopping Speed (mm/min)[>100]	100
51	SP_Speed when exit at following tapping mode(rpm)[>1]	1

52	SP_Reverse Backlash Compensation when tapping(Pulse)	1
53	Advance Retired Value before reverse rotation of following tapping(um)[10-5000]	1
54	Retired Speed when tapping (mm/min)[>=60]	1
58	Beginning Speed when hardware limitation (mm/min)	1.000
60	Activate G01/G02/G03 smooth running(1: No, 60:Yes)	60
61	Time Constant for G01/G02/G03 smooth running [2-50]	50
101	X_Beginning Speed (mm/min)[>1]	500
102	Y_Beginning Speed (mm/min)[>1]	500
103	Z_Beginning Speed (mm/min)[>1]	500
104	A_Beginning Speed (mm/min)[>1]	500
105	B_Beginning Speed (mm/min)[>1]	500
111	X_Max Speed Increment (mm/min)[>1]	500
112	Y_Max Speed Increment (mm/min)[>1]	500
113	Z_Max Speed Increment (mm/min)[>1]	500
114	A_Max Speed Increment (mm/min)[>1]	500
115	B_Max Speed Increment (mm/min)[>1]	500
200	Coherent movement is valid for G00[1:No,16:Yes]	1
201	Wait SP_Speed smooth when threading	1

Explanation about Speed Parameter:

1,G00 Speed of X-axis (mm/min) 2,G00 Speed of Y-axis (mm/min)

3,G00 Speed of Z-axis (mm/min) 4,G00 Speed of A-axis (mm/min)

It is rapid speed(also speed of G00) of X/Z axis,Max is 240000(unit:mm/min)

Attention: the value depends on machine configuration,set wrong is very easy to trouble machine tool & accident.

5,Manual Max Feeding Speed(mm/min)

It is the max feeding speed in the condition of Manual , Unit:mm/min.

Attention: reference speed=G00 speed*0.5 ,in order to make sure safe.

6,Auto Max Feeding Speed(mm/min)

It is the max of feeding speed in Auto ,Max is 30000.Unit:mm/min. This speed could faster than G00 speed in order to each axis can reach at max speed when linkage as it only limit integrated speed.

7,Default Speed of G01/G02/G03 (mm/min)

It is the default speed of G01/G02/G03 when the speed of first interpolation code (G01/G02/G03) isn't specified in Auto-running. Max:5000 (unit:mm/min)

8,Running Speed at Simulation Mode (mm/min)

It is running speed at the mode of Simulation. (press "Simulate",is dry run mode) Max:240000. (unit:mm/min)

9,Manual Feeding Speed(mm/min)

It is the speed of feeding axis in Manual.Range:< max feeding speed

Attention:in Manual mode, press "F" key ,can set the parameter directly.

10,Manual Spindle Speed (rpm)

It is set for speed of spindle in mode of manual. Unit:rpm.

Attention:in Manual,press "S" key ,can set the parameter directly.

13,Limit G01/G02/G03 Speed of each axis [1:Yes, 0:No]

It is set for whether limit speed of each axis when G1/G2/G3 interpolating.

14,Max Speed of X_G1/G2/G3 (mm/min)

It is for the Max running speed of X-axis when G1/G2/G3 interpolation.

15,Max Speed of Y_G1/G2/G3 (mm/min)

It is for the Max running speed of Y-axis when G1/G2/G3 interpolation.

16,Max Speed of X_G1/G2/G3 (mm/min)

It is for the Max running speed of Z-axis when G1/G2/G3 interpolation.

17,Max Speed of Z_G1/G2/G3 (mm/min)

It is for the Max running speed of A-axis when G1/G2/G3 interpolation.

18,X_Acceleration/Deceleration Constant [1～99999]

It is time constant of X-axis acc/dec-eleration,the bigger it is ,the faster the ace/dec-eleration is.

Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

19,Y_Acceleration/Deceleration Constant [1～99999]

It is time constant of Y-axis acc/dec-eleration,the bigger it is ,the faster the ace/dec-eleration is.

Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

20,Z_Acceleration/Deceleration Constant [1～99999]

It is time constant of Z-axis acc/dec-eleration, the bigger it is ,the faster the ace/dec-eleration is.

Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

21,X_Acceleration/Deceleration Constant [1～99999]

It is time constant of A-axis acc/dec-eleration,the bigger it is ,the faster the ace/dec-eleration is.

Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

22,Acceleration/Deceleration Constant When Auto Run [1-500]

It is for set constant of acc/dec-eleration in auto.the range is 1-500.It is mainly for distinguish Auto and Manual,only the difference is too much,set it is effective.

23,Handwheel_Acc/Dec-eleration Constant [500--32000]

It is for set constant of acc/dec-eleration of Handwheel.the range is 500-32000.

24,Handwheel_Acc/Dec-eleration Constant when dry [>500]

It is for set constant of acc/dec-eleration when handwheel start program. the range is 500-32000.when the value is less than 500,it is invalid.

25,G00 Speed when dry (mm/min) [>10]

It is the speed of G00 when when handwheel start program for simulate. it is invalid when the value is less than 10.

26,X_Max Speed with Handwheel (mm/min)

It is for limit max speed of X-axis when use handwheel in manual.

Attention:it is valid when >100,otherwise invalid.And suggest don't over 4000.

27,Y_Max Speed with Handwheel (mm/min)

It is for limit max speed of Y-axis when use handwheel in manual.

Attention:it is valid when >100,otherwise invalid.And suggest don't over 4000.

28,Z_Max Speed with Handwheel (mm/min)

It is for limit max speed of Z-axis when use handwheel in manual.

Attention:it is valid when >100,otherwise invalid.And suggest don't over 4000.

29,A_Max Speed with Handwheel (mm/min)

It is for limit max speed of A-axis when use handwheel in manual.

Attention:it is valid when >100,otherwise invalid.And suggest don't over 4000.

30,Type of Acceleration/Deceleration [0 mean line,8 mean curve]

It sets type of acc/dec-eleration.set 0 means line type.set 8 means curve type.

Attention:In normal condition,set line type in servo system;set curve type in step system.

31,Initial Acc/Dec-eleration Constant when curve [>=10]

It is initial acce/dec-eleration constant when P27 set curve type.Range: >=10.

32,Quadratic Acce/Dece-leration Constant when curve [>=10]

It is quadratic constant of acceleration/deceleration when P27 set curve type. Range >=10.

33,Max Acce/Dece-leration Constant when curve [>=500]

It is Max acceleration/deceleration constant when P27 set curve type.

It is valid when >=500,otherwise the ace/dec-eleration constant is with line type of each axis.

34,X_Homing Speed (mm/min)

It is homing speed of X-axis.Unit:mm/min. the range is less than X_G00 speed.

35,X_Speed for detect Z0 signal (mm/min)

It is speed for check Z0 pulse signal after X-axis reach at homing switch. Unit:mm/min. the range is 20-500.

note:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

36,Y_Homing Speed (mm/min)

It is homing speed of X-axis.Unit:mm/min. the range is less than Y_G00 speed.

37,Y_Speed for detect Z0 signal (mm/min)

It is speed for check Z0 pulse signal after Y-axis reach at homing switch. Unit:mm/min. the range is 20-500.

note:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

38,Z_Homing Speed (mm/min)

It is homing speed of Z-axis.Unit:mm/min. the range is less than Z_G00 speed.

39,Z_Speed for detect Z0 signal (mm/min)

It is speed for check Z0 pulse signal after Z-axis reach at homing switch. Unit:mm/min. the range is 20-500.

Note:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

40,A_Homing Speed (mm/min)

It is homing speed of A-axis.Unit:mm/min. the range is less than X_G00 speed.

41,A_Speed for detect Z0 signal (mm/min)

It is speed for check Z0 pulse signal after A-axis reach at homing switch. Unit:mm/min. the range is 20-500.

note:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

42,Max Speed of 1st Spindle (rpm)

It is max speed of 1st spindle, also at 1st gear,M41 output for 1st gear, it is also the speed when PIN25_CN3 plug output analog voltage is 10V(Default condition).

43,Max Speed of Spindle at 2nd gear (rpm)

It is max speed of spindle at 2nd gear,M42 output for 2nd gear, it is also the speed when PIN25_CN3 plug output analog voltage is 10V at M42.

44,Max Speed of Spindle at 3rd gear (rpm)

It is max speed of spindle at 3rd gear,M43 output for 3rd gear, it is also the speed when PIN25_CN3 plug output analog voltage is 10V at M43.

45,Max Speed of Spindle at 4th gear (rpm)

It is max speed of spindle at 4th gear,M44 output for 4th gear, it is also the speed when PIN25_CN3 plug output analog voltage is 10V at M44.

46,Max Speed of 2nd Spindle (rpm)

It is the max speed of 2nd spindle,it is also the speed when PIN25_CN10 plug output analog

voltage is 10V. Specified by “SS#” , unit is rpm.

47,Compensation Mode of Arc Backlash (0 mean A; 8 mean B)

It is compensation mode for arc reverse backlash.

0 means A compensation mode, which is that the bigger the reverse backlash is ,the faster the compensation speed is , in order to ensure tool don’t exit pause condition. the compensation speed is less than 1000mm/min.

8 means B compensation mode , which is that the compensation speed is set by related parameters in the following.

+4: means when arc programming, IJK is the coordinate value from end point to center. In the original value of P41 plus 4(Eg.: P41=0 +4= 4) means that the IJK of G02/G03 is the coordinate value from end point to center, otherwise IJK of G02/G03 is the coordinate value from starting point to center.

48,Compensation Speed in B Mode (mm/min)

It is the compensate speed in B compensation mode.unit:mm/min.

48-1,Beginning Compensation Speed in B mode (mm/min)

It is beginning compensate speed in B compensation mode.it is valid when it >10.

48-2,Acce/Dece-leration Constant in B mode (mm/min/s)

It is the acceleration/deceleration constant in B compensation mode (also P41=8/12) . Range: >=10.

49,Activate Speed Processing Function[1:Yes ; 0:No]

It is set for if activate speed processing function, 1 means yes, activate speed processing function, 0 means no activate the function.

50,Handwheel_Stopping Speed (mm/min)[>100]

It is the speed when handwheel stop. the bigger it is ,the faster handwheel stop.

51,SP_Speed when exit at following tapping mode

It is min speed before spindle reverse rotation when tapping.

52,SP_Reverse Backlash Compensation when tapping

It is reverse backlash compensation value before spindle reverse rotation when tapping. Unit: Pulse

53,Advance Retired Value before reverse rotation of following tapping

It is advance retired value before spindle reverse rotation when tapping. Unit:um. Range:10-5000

54,Retired Speed when tapping (mm/min)[>=60]

It is speed when spindle retired during tapping. Unit: mm/min

58,Beginning Speed when hardware limitation (mm/min)

It is beginning speed that motor touch hardware limitation switch. When CNC system is configured with servo, no need to set beginning speed , set to 1 normally.

60,Activate G01/G02/G03 smooth running(1:is No,60:is Yes)

It is set for if activate the function of smooth running for G01/G02/G03 at processing program. Set to 60 means activate the function, 1 means no.

61,Time Constant for G01/G02/G03 smooth running [2-50]

It is the time constant when CNC system activate function of G01/G02/G03 smooth running,also P60=60. Range: 2-50.

101,X_Beginning Speed (mm/min)[>1]
102,Y_Beginning Speed (mm/min)[>1]
103,Z_Beginning Speed (mm/min)[>1]
104,A_Beginning Speed (mm/min)[>1]
105,B_Beginning Speed (mm/min)[>1]

It is beginning speed of feeding axis when acceleration/deceleration.when it is smaller than acceleration/deceleration, accelerate/decelerate of speed start from the beginning feed speed. when it is bigger than acceleration/deceleration, the speed reached at setting speed directly. Unit: mm/min.

Attention:Generally,stepper system<=100,servo system<=500.

111,X_Max Speed Increment (mm/min)[>1]
112,Y_Max Speed Increment (mm/min)[>1]
113,Z_Max Speed Increment (mm/min)[>1]
114,A_Max Speed Increment (mm/min)[>1]
115,B_Max Speed Increment (mm/min)[>1]

It is max speed increment when multi-axial running track-interpolation. Also max changing value of speed.

Example:when it is 300,the speed of X axis(multi-axial track-interpolation)up from F800 to F1600,800(=1600-800)>300,so the process is up from F800 to F1100,and then F1600.

200,Coherent movement is valid for G00[1 is No,16 is Yes]

It is set for that if coherent movement is valid for G00. 16: yes, it is valid for G00. 1: No, it is invalid for G00.

210,Wait SP_Speed smooth when tapping[0:No, 1:Yes]

It is set for that if wait speed of spindle is smooth when tapping. 1: yes, wait speed of spindle is smooth before threading. 0: No, don’t wait.

2.3 Axis parameter

P	Axis Parameter	Ex-Value
1	Switch Type for Feed-Rate [0: Key, 1: Band Switch]	0
2	Switch Type for SP-Rate [0: Key, 1: Band Switch]	0
3	Max Travel in X_Negative direction (mm)	-9999.000
4	Max Travel in X_Positive direction (mm)	9999.000
5	Max Travel in Y_Negative direction (mm)	-9999.000
6	Max Travel in Y_Positive direction (mm)	9999.000
7	Max Travel in Z_Negative direction (mm)	-9999.000
8	Max Travel in Z_Positive direction (mm)	9999.000
9	Max Travel in A_Negative direction (mm)	-9999.000
10	Max Travel in A_Positive direction (mm)	9999.000
11	SP_Braking Time (10ms)	150
12	SP_Braking is Long Signal [0: No,1: Yes]	0
13	Soft-Limitation is valid [0:Yes, 1: No][D2X;D3C(Y);D4Z;D5A;D6B]	00000001
14	X_Reverse Backlash Compensation (um)	0
15	Y_Reverse Backlash Compensation (um)	0
16	Z_Reverse Backlash Compensation (um)	0
17	A_Reverse Backlash Compensation (um)	0
18	X_Direction [1:normal, 0: Reverse]	0
19	Y_Direction [1:normal, 0: Reverse]	0
20	Z_Direction [1:normal, 0: Reverse]	0
21	A_Direction [1:normal, 0: Reverse]	0
22	Using Electron Gear Ratio for Feeding Axes [0:Yes, 1:No]	0
23	Numerator of X_Electron Gear	1
24	Denominator of X_Electron Gear	1
25	Numerator of Y_Electron Gear	1
26	Denominator of Y_Electron Gear	1
27	Numerator of Z_Electron Gear	1
28	Denominator of Z_Electron Gear	1
29	Numerator of A_Electron Gear	1
30	Denominator of A_Electron Gear	1
31	Type of Limit Switch in Positive Direction[0:NO type, 1:NC type]	0
32	Type of Limit Switch in Negative Direction[0:NO type, 1:NC type]	0
33	Type of Home [D3X;D4(C)Y;D5Z;D6A;0:Switch;1:float Zero]	01111011
34	X_Machine Coordinate of float zero point	
35	Y_Machine Coordinate of float zero point	
36	Z_Machine Coordinate of float zero point	
37	A_Machine Coordinate of float zero point	
38	Grade of Homing[1:No,0:Prompt,8:Compulsion,9:Super compulsion]	1
39	Mode of Homing	1
40	Direction of Homing[D2:X,D3:C(Y),D4:Z,D5:A,D6:B]	0000000100000001
41	Type of Switchs for Homing	00000000
42	Range of Detect Z0 in X axis	100
43	Range of Detect Z0 in Y axis	100
44	Range of Detect Z0 in Z axis	100
45	Range of Detect Z0 in A axis	100
46	Offset after homing in X axis	0
47	Offset after homing in Y axis	0
48	Offset after homing in Z axis	0

49	Offset after homing in A axis	0
50	Spindle is rotating when shift gear[1:Yes, 0:No]	0
51	Rotating Speed of Spindle when shift gear(1/100rpm)	1000
52	Rotating Direction of Spindle when shift gear[0:CW, 1:CCW]	1
53	Pause Time of Spindle when shift gear (10ms)	10
54	Braking Time of Spindle when shift gear (10ms)	10
55	Delay time between reset M03/M04 & set M05 (10ms)	0
56	Detect SP_Position Feedback [0: No,1: Yes]	1
57	Pulses Per Revolution of Spindle	4096
58	SP_CCW is Long Signal [8: No, other: Yes]	0
68	Delay time when feeding axes shift direction	0
80	X/Z axis is Rotating Axis	00000001
81	Mode of Y(C) Axis [0:Rotating Axis, 1: Linear Axis]	
82	Workpiece coordinate when Y is rotating axis	
83	Machine Coordinate when Y is rotating axis	
100	System Inner Parameter	1
101	Mode of A Axis [0:Rotating Axis, 1: Linear Axis]	1
102	Base when A axis is rotating axis	-9999.000
300	System Inner Parameter	
301	Mode of B Axis [0:Rotating Axis, 1: Linear Axis]	1
302	Base when B axis is rotating axis	0
303	Name of 5th Axis [0:B, 1:C]	0
304	B_Direction [1:normal, 0: Reverse]	0
305	Numerator of B_Electronic Gear	1
306	Denominator of B_Electronic Gear	1
307	B_Reverse Backlash Compensation (um)	0
308	G00 Speed of B Axis	4000.000
309	Max Speed of B_G1G2G3	2000
310	B_Acceleration/Deceleration Constant	50000
311	B_Max Speed with Handwheel (mm/min)	2000
312	Homing Speed in B Positive Direction (mm/min)	2500.000
313	Homing Speed in B Negative Direction (mm/min)	250
314	Range of detect Z0 in B axis	60
315	Offset after homing in B axis	0
316	Max Travel in B-Negative Direction	-9999.000
317	Max Travel in B-Positive Direction	9999.000
318	B_Machine Coordinate of float zero point	
400	Detect Position Feedback of Spindle When ATC(0:No,1:Yes)	
401	Detecting Angle when SP-Orientation	
402	Allowable error angle when SP-Orientation	
404	SP_Direction when position control mode	0
405	Using Electronic Gear Ratio for Spindle [0:Yes, 1:No]	0
406	Numerator of SP_Electronic Gear Ratio in Low Gear	4096
407	Denominator of SP_Electronic Gear Ratio in Low Gear	36000
408	Numerator of SP_Electronic Gear Ratio in High Gear	4096
409	Denominator of SP_Electronic Gear Ratio in High Gear	360000
410	Coordinate Axis when spindle do tapping	94
411	Control Mode of Tapping [2:Following, 3: Interpolation]	3
412	Teeth of SP_Motor (<P413)	1
413	Teeth of SP_Encoder (>P412)	1
414	Follow-Up of A Axis[7:X, 8:Y, 9:Z]	1

Note: System inner parameters cannot be changed.

Explanation about Axis Parameter:

1,Switch Type for Feed-Rate [0: Key, 1: Band Switch]

It is set switch type of Feed-Rate, Rate of Feeding axes.

0: Keys of "Feed Rate+" & "Feed Rate-" in Panel for adjust rate of feeding speed;

1: External Band switch for Feed-Rate. Port for external band switch is at CN11 plug, Pins are VDK0,VDK1,VDK2,VDK3. Total are 16 gears

2,Switch Type for SP-Rate [0: Key, 1: Band Switch]

It is set switch type of SP-Rate, Rate of SP_Speed.

0: Keys of "SP Rate+" & "SP Rate-" in Panel for adjust rate of spindle speed;

1: External Band switch for SP-Rate. Port for external band switch is at CN11 plug, Pins are VDS0,VDS1,VDS2,VDS3. Total are 16 gears

3,Max Travel in X_Negative direction (mm)

It is max travel in negative direction of X axis when soft-limitation, which is based on machine coordinate system.

4,Max Travel in X_Positive direction (mm)

It is max travel in positive direction of X axis when soft-limitation, which is based on machine coordinate system.

5,Max Travel in Y_Negative direction (mm)

It is max travel in negative direction of Y axis when soft-limitation, which is based on machine coordinate system.

6,Max Travel in Y_Positive direction (mm)

It is max travel in positive direction of Y axis when soft-limitation, which is based on machine coordinate system.

7,Max Travel in Z_Negative direction (mm)

It is max travel in negative direction of Z axis when soft-limitation, which is based on machine coordinate system.

8,Max Travel in Z_Positive direction (mm)

It is max travel in positive direction of Z axis when soft-limitation, which is based on machine coordinate system.

9,Max Travel in A_Negative direction (mm)

It is max travel in negative direction of X axis when soft-limitation, which is based on machine coordinate system.

10,Max Travel in A_Positive direction (mm)

It is max travel in positive direction of X axis when soft-limitation, which is based on machine coordinate system.

11,SP_Braking Time (10ms)

It is the braking time of spindle,also holding time of output M05. the shorter it is,the faster the brake is. Unit: 10ms.

12,SP_Braking is Long Signal [0: No,1: Yes]

It is set for signal mode of SP_Braking. 1: Long signal, 0:short signal. It depends on braking mode of spindle system.

13,Soft-Limitation is valid [0:Yes, 1: No]

It is bit parameter, set for if soft-limitation is valid of each feeding axis.

Bit	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1
Axis	-	B	A	Z	Y	X	-	-

0: means soft-limitation is valid , 1: means invalid

Example:the soft limit of X-axis is valid, P11= 00000100.

14,X_Reverse Backlash Compensation (um)

It is the value of reverse backlash compensation of X-axis. When direction of X-axis movement is changed ,system will make compensation with the value automatically. Unit: um

15,Y_Reverse Backlash Compensation (um)

It is the value of reverse backlash compensation of Y-axis. When direction of Y-axis movement is changed ,system will make compensation with the value automatically. Unit: um

16,Z_Reverse Backlash Compensation (um)

It is the value of reverse backlash compensation of Z-axis.When direction of Z-axis movement is reversed ,system will make compensation with the value automatically. Unit: um

17,A_Reverse Backlash Compensation (um)

It is the value of reverse backlash compensation of A-axis, When direction of A-axis movement is changed ,system will make compensation with the value automatically. Unit: um

18,X_Direction [1:normal, 0: Reverse]

It is for set the direction of X-axis. 1: Direction of X-axis is same to direction of code; 0: Direction of X-axis is opposite to direction of code.

19,Y_Direction [1:normal, 0: Reverse]

It is for set the direction of Y-axis. 1: Direction of Y-axis is same to direction of code; 0: Direction of Y-axis is opposite to direction of code.

20,Z_Direction [1:normal, 0: Reverse]

It is for set the direction of Z-axis. 1: Direction of Z-axis is same to direction of code; 0: Direction of Z-axis is opposite to direction of code.

21,A_Direction [1:normal, 0: Reverse]

It is for set the direction of A-axis. 1: Direction of A-axis is same to direction of code; 0: Direction of A-axis is opposite to direction of code.

22,Using Electron Gear Ratio for Feeding Axes [0:Yes, 1:No]

It is for whether using the electron gear ratio for feeding axis. 0: yes,using electron gear, 1: No, don't using electron gear.

23,Numerator of X_Electron Gear (1-999999)

It is Numerator of X-axis's electron gear ratio.(X_CMR) Range: 1-999999.

24,Denominator of X_Electron Gear (1-999999)

It is Denominator of X-axis's electron gear ratio.(X_CMD) Range: 1-999999.

25,Numerator of Y_Electron Gear (1-999999)

It is Numerator of Y-axis's electron gear ratio.(Y_CMR) Range: 1-999999.

26,Denominator of Y_Electron Gear (1-999999)
It is Denominator of Y-axis's electron gear ratio.(Y_CMD) Range: 1-999999.
27,Numerator of Z_Electron Gear (1-999999)
It is Numerator of Z-axis's electron gear ratio. (Z_CMR) Range: 1-999999.

28,Denominator of Z_Electron Gear (1-999999)
It is Denominator of Z-axis's electron gear ratio.(Z_CMD) Range: 1-999999.

29,Numerator of A_Electron Gear (1-999999)
It is Numerator of A-axis's electron gear ratio.(A_CMR) Range: 1-999999.

30,Denominator of A_Electron Gear (1-999999)
It is Denominator of A-axis's electron gear ratio.(A_CMD) Range: 1-999999.

Algorithm of P17-P20 & P105/P106 & P204/P205 parameters
Effective Range: 1-999999
Unit:non
User:Upon operating administrators
Initialization:1
Effective time:Immediately
Explain:
When lead screws with different screw pitches are configured with motors of various step angles,or with servo motors of different pulse number per round,or connections are realized through different gears,the programmed values can remain consistent with the actual moved distance by setting the parameter of the electronic gear ration of the system.

$$\text{Electron Gear Ratio} = \frac{\text{Numerator}}{\text{Denominator}} = \frac{\text{CMR}}{\text{CMD}} = \frac{P}{L*1000}$$

CMR:Numerator of gear ratio
CMD:Denominator of gear ratio
P: pulse number per motor round
L: Moved distance per motor round(mm)
The value of CMD/CMR is the pulse equivalent,which tells the moved distance per pulse ,with its unit as 0.001mm.
Example1: The motor rotates one circle very 5000 pulses,after which the machine tool moves 5mm,then:
CMR/CMD = 5000 / (5 * 1000) = 1/ 1
That is to say,we can set the values as :CMR=1,CMD=1.
Here ,the pulse equivalent is 0.001mm.
Example2: The motor rotates one circle very 5000 pulses,after which the machine tool moves 10mm.
CMR/CMD = 5000 / (10 * 1000) = 1 / 2
That is to say,we can set the values as :CMR=1,CMD=2.
Here ,the pulse equivalent is 0.002mm.

31,Type of Limit Switch in Positive Direction[0:NO type, 1:NC type]
It is set type of limit switch in positive direction, also type of switch that is connected to +L,PIN16_CN3 plug. 0:NO Type, 1: NC Type.

32,Type of Limit Switch in Negative Direction[0:NO type, 1:NC type]
It is set type of limit switch in negative direction, also type of switch that is connected to -L,PIN15_CN3 plug. 0:NO Type, 1: NC Type.

33,Type of Home [D3X;D4Y;D5Z;D6A;D7B 0:Switch; 1:float Zero]

It is set type of home. bit parameter. Each axis set alone.

Bit	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1
Axis	B	A	Z	Y	X	-	-	-

0: Switch/Sensor for home, 1:Float Zero point for home.

Example:Only Home of X-axis is float zero point, P23=00001001.

34,X_Machine Coordinate of float zero point

It is set the machine coordinate value of X-axis based on float zero point. The value is distance between current position of machine tool & float zero point.

35,Y_Machine Coordinate of float zero point

It is set the machine coordinate value of Y-axis based on float zero point. The value is distance between current position of machine tool & float zero point.

36,Z_Machine Coordinate of float zero point

It is set the machine coordinate value of Z-axis based on float zero point. The value is distance between current position of machine tool & float zero point.

37,A_Machine Coordinate of float zero point

It is set the machine coordinate value of A-axis based on float zero point. The value is distance between current position of machine tool & float zero point.

38,Grade of Homing[1:No need, 0:Prompt, 8:Compulsion, 9:Super compulsion]

It set the grade of homing for feeding axis.there is 4 grades as follow:

1 : No need. When system boots every time,no prompt and no limitation;

0 : Prompt. After system boot every time,there is a prompted box for homing , and then there aren't any limitation about homing;

8 : Compulsion. When system boots every time, there will a prompted box for homing. And then, if system don't homing successfully, it will hint "feed axis don't go home" before running program ,and don't run processing program;

9 : Super compulsion. When system boots every time, there will a prompted box for homing. And then,if system don't homing successfully,it will hints "feed axis don't go home" at each operations ,and feeding axes don't move.

39,Mode of Homing

It set mode of homing. There are 4 kinds of mode about homing.

0 : Homing after hit homing switch, move in reverse direction until homing switch is off, then detect Z0 signal of Encoder of servo motor.

1 : Homing after hit homing switch, move in reverse direction until homing switch is off.

2 : Homing after hit homing switch, move forward until homing switch is off, then detect Z0 signal of encoder of servo motor.

Other: Homing after hit homing switch,move forward until homing switch is off.

40,Direction of Homing [D2:X,D3:C(Y),D4:Z,D5:A]

It sets the direction & sequence of homing for each axis. Bit parameter.

Bit	D15	D14	D13	D12	D11	D10	D9	D8	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1	0	0	0	0	0	0	0	1
Axis	-	-	-	-	-	-	-	-	-	B	A	Z	Y	X	-	-

0:Homing in positive direction, 1:Homing in negative direction, D8: set priority of X&Z-axis go home.1 means Z-axis first,0 means X-axis first.

41,Type of Switch for Homing [D0X;D1C(Y);D2Z;D3A;D4B; 1:NC ; 0:NO]

It set the type of switch for homing. Bit parameter.

Bit	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1
Axis	-	-	-	B	A	Z	C(Y)	X

1: NC type; 0: NO type.D7=1:Manual/Auto shift automatically ;

Example:If X&Z axis are NC switch,the bit parameter is 000000101.

42,Range of Detect Z0 in X axis (unit:100um)

It is range that system can detect Z0 signal of encoder in X direction.

Attention:the value must be less than the length of one rev,otherwise homing failure.

43,Range of Detect Z0 in Y axis (unit:100um)

It is range that system can detect Z0 signal of encoder in Y direction.

Attention:the value must be less than the length of one rev,otherwise homing failure.

44,Range of Detect Z0 in Z axis (unit:100um)

It is range that system can detect Z0 signal of encoder in Z direction.

Attention:the value must be less than the length of one rev,otherwise homing failure.

45,Range of Detect Z0 in A axis (unit:100um)

It is range that system can detect Z0 signal of encoder in A direction.

Attention:the value must be less than the length of one rev,otherwise homing failure.

46,Offset after homing in X axis (unit:10um,-9999~+9999)

It is offset that X-axis after homing. Move with G00 speed. Unit: 0.01mm.

47,Offset after homing in Y axis (unit:10um,-9999~+9999)

It is offset that Y-axis after homing. Move with G00 speed. Unit: 0.01mm.

48,Offset after homing in Z axis (unit:10um,-9999~+9999)

It is offset that Z-axis after homing. Move with G00 speed. Unit: 0.01mm

49,Offset after homing in X axis (unit:10um,-9999~+9999)

It is offset that A-axis after homing. Move with G00 speed. Unit: 0.01mm.

50,Spindle is rotating when shift gear [1:Yes, 0:No]

It set if spindle is rotating when spindle shift gears.1:Yes, SP is rotating, 0: No.

51,Rotating Speed of Spindle when shift gear(1/100rpm)

It is rotating speed of spindle when spindle shift gear & P51=1.

52,Rotating Direction of Spindle when shift gear [0:CW,1:CCW]

It is rotation direction of spindle when spindle shift gear. 0: CW, output M03; 1:CCW, output M04.

53,Braking Time of Spindle when shift gear (10ms)

It is braking time of spindle when spindle shift gear. Unit: 10ms.

54,Delay time between reset M03/M04 & set M05 (10ms)

It is delay time before output M05 ,and after reset M03/M04. Unit:10ms.

55,Spindle stop time(unit:10ms)
It is the delay time between cancel M03/M04 and boot M05.unit:10ms.

56,Detect SP_Position Feedback [0: No,1: Yes]
It is for whether the system detect position feedback signal of spindle by SP_encoder. 1: detect, 0: No detect.
It is used for open that display of spindle real speed & some functions related with SP_Speed which must be on condition of transmission ratio is 1:1 between SP_encoder & spindle motor.

57,Pulses Per Revolution of Spindle
It is pulses per revolution of spindle. Pulses= (Resolution of SP-encoder) * 4.

58,SP_CCW is Long Signal [8: No, other: Yes]
It is set for signal mode of SP_Braking. 1: Long signal, 0:short signal. It depends on braking mode of spindle system.

68,Delay time when feeding axes shift direction(ms)
It sets delay time when feeding axes(XYZA) change direction, unit: ms.
80,Mode of X&Z axis
It is bit parameter, Each bit have its related function. 1: Valid, 0: Invalid.
D2:Z axis based on Workpiece coordinate system;D3:X axis based on Workpiece coordinate system; D4:Z axis based on Machine coordinate system; D5:X axis based on Machine coordinate system. D6:Z axis is rotation axis; D7:X axis is rotation axis.

Bit	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1
Func	XR	ZR	XM	ZM	XW	ZW	-	-

81,Mode of Y(C) Axis [0:Rotating Axis, 1: Linear Axis]
It is set for the mode of Y(C)-axis, 0: Rotating axis , 1: Linear axis.

100,System Inner Parameter
Inner parameter of system, cannot be altered.

101,Mode of A Axis [0:Rotating Axis, 1: Linear Axis]
It is set for the mode of A-axis, 0: Rotating axis , 1: Linear axis.

102,Base when A axis is rotating axis
It is set the base of A-axis when it is rotating axis. 0:Null, 1:Based on Absolute Coordinate, 2: Based on Machine Coordinate, 3: Both.

300,System Inner Parameter
Inner parameter of system, cannot be altered.

301,Mode of B Axis [0:Rotating Axis, 1: Linear Axis]
It is set for the mode of B-axis, 0: Rotating axis , 1: Linear axis.

302,Base when B axis is rotating axis
It is set the base of B-axis when it is rotating axis. 0:Null, 1:Based on Absolute Coordinate, 2: Based on Machine Coordinate, 3: Both.

303,Name of 5th Axis [0:B, 1:C]
It is set for name of 5th axis, 0: set to B , display & programming with “B”, normally when 5th

axis is linear axis; 1: set to C, display & programming with “C”, normally when 5th axis is rotating axis.

304,B_Direction [1:normal, 0: Reverse]

It is for set the direction of B-axis. 1: Direction of B-axis is same to direction of code; 0: Direction of B-axis is opposite to direction of code.

305,Numerator of B_Electronic Gear

It is Numerator of B-axis’s electron gear ratio. (B_CMR) Range: 1-999999.

306,Denominator of A_Electronic Gear

It is Denominator of A-axis’s electron gear ratio.(B_CMD) Range: 1-999999.

307,B_Reverse Backlash Compensation (um)

It is the value of reverse backlash compensation of B-axis.When direction of B-axis movement is changed ,system will make compensation with the value automatically. Unit: um

308,G00 Speed of B Axis

It is rapid speed(also speed of G00) of B axis,Max is 240000(unit:mm/min)

Attention: the value depends on machine configuration,set wrong is very easy to trouble machine tool & accident.

309,Max Speed of B_G1G2G3

It is for the Max running speed of B-axis when G1/G2/G3 interpolation.

310,B_Acceleration/Deceleration Constant

It is time constant of B-axis acce/dece-leration,the bigger it is ,the faster the ace/dec-eleration is.

Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

311,B_Max Speed with Handwheel (mm/min)

It is for limit max speed of B-axis when use handwheel in manual.

Attention:it is valid when >100,otherwise invalid.And suggest don’t over 4000.

312,B_Homing Speed (mm/min)

It is homing speed of B-axis .Unit:mm/min. the range is less than B_G00 speed.

313,B_Speed for Detect Z0 signal (mm/min)

It is speed for check Z0 pulse signal after B-axis reach at homing switch.

314,Range of detect Z0 in B axis

It is range that system can detect Z0 signal of encoder in B direction.

Attention:the value must be less than the length of one rev,otherwise homing failure.

315,Offset after homing in B axis

It is offset that B-axis after homing. Move with G00 speed. Unit: 0.01mm

316,Max Travel in B-Negative Direction

It is max travel in negative direction of B axis when soft-limitation, which is based on machine coordinate system.

317,Max Travel in B-Positive Direction

It is max travel in positive direction of B axis when soft-limitation, which is based on machine coordinate system.

318,B_Machine Coordinate of float zero point

It is set the machine coordinate value of B-axis based on float zero point. The value is distance between current position of machine tool & float zero point.

404,SP_Direction when position control mode

It is the direction of spindle motor,o means reverse,1 mean normal.

405,Using Electronic Gear Ratio for Spindle [0:Yes, 1:No]

It is for whether the spindle use electron gear.

406,Numerator of SP_Electronic Gear Ratio in Low Gear (1-999999)

It is the numerator of SP-axis's electron low gear in low gear.

407,Denominator of SP_Electronic Gear Ratio in Low Gear (1-999999)

It is the denominator of SP-axis's electron low gear in low gear.

408,Numerator of SP_Electronic Gear Ratio in High Gear (1-999999)

It is the numerator of SP-axis's electron low gear in high gear.

409,Denominator of SP_Electronic Gear Ratio in High Gear (1-999999)

It is the denominator of SP-axis's electron low gear in high gear.

410,Coordinate Axis when spindle do tapping[91 X,92 Y/C,93 Z,94 A,95 B]

It is the axis that be use for spindle when interpolation tap.

411,Control Mode of Tapping [0:Following, 1: Interpolation]
[2 follow encode;3 interpolation to SP]

It is control mode of interpolation tap.

412,Teeth of SP_Motor (<P413)

It is tooth number of spindle.it <=P413.

413,Teeth of SP_Encoder (>P412)

It is tooth number of SP-encoder,it >=P412.

Attention:the tooth number of spindle must be not more than the tooth number of SP-encoder,when less,it need to install our company's adapter plate.

414,Follow-Up of A Axis[7:X, 8:Y, 9:Z]

It is set the function of follow up of A-axis. Set to 7: A-axis follow up with X-axis; Set to 8: A-axis follow up with Y-axis ; Set to 9: A-axis follow up with Z-axis. A-axis follow up on condition of both Manual & Auto.

Special Cautions:

1.Only when CNC controller is configured with related feeding axis, and there are related parameters sets for related feeding axis, such as C axis & A-axis.

2.About bit parameters, if some bits are don't specified functions for feeding axis, please keep same to ex-factory set, which should be important for inner system, otherwise it will affect normal operations of CNC system.

2.4 Tool parameter

P	Tool Parameter	Ex-Value
1	Mode of Setup Radius C Compensation	0
2	Mode of Cancel Radius C Compensation	0
10	Change tool function[1 Yes, 0 No]	
11	Go home axis when change tool[3 mean ZX;4 mean ZY;5 mean ZXY;Others means Z]	
12	Go home reference point when change tool[1,2,3,4]	
13	Check SP fix position(1 Yes, 0 No)	
14	SP go to fix position when change tool[1 Yes, 0 No]	
15	Tools storeroom type[0 Revolve machine hand,1 line tools,2 circle plate]	
16	Special Tools storeroom[0:Standard , 16: Special Circle]	
17	Z-axis coordinate when change Circle plate tool storeroom	
18	Z-axis speed when change Circle plate tool storeroom	
19	SP chuck close delay(0.1s)	
20	SP Chuck loose delay(0.1s)	
21	Tool cover control delay(ms)	
22	22,Does lock for Z-axis move & Tools storeroom control(0 mean no,1 mean Yes)	
23	Does check Z-axis position when Tools storeroom go forward instruction M71 is valid	
24	Maximum tool number(setup this parameter,must initialize tool table)	
25	Set up T08 signal[rise “1000+input number”,drop “2000+input number”]	
26	Set up M61signal[1000+output number]	
27	Set up M63 signal[1000+output number]	
28	Set up M71 signal[1000+output number]	
29	Set up M42 signal[1000+output number]	
30	Set up Y24 signal[1000+output number]	
32	Filtering for Position Signal or WAT Signal	1287
70	Servo tools go home speed(degree/min)	
71	Servo tools go home reverse speed(degree/min)	
72	Servo tools go home offset(0.001degree)	
73	Servo change tool speed(degree/min)	
74	Servo tools axis(1=A; 2=B; 3=C)	
75	Servo tools acceleration(degree/min/s)	
76	Servo tools go home max distance(0.001degree)	
77	Servo tools go home check motor zero(0: is No. 1: is Yes)	
78	Servo tools revolve one tool pulse	
79	Servo tools go home current tool No.	
100	Reference point 1 X(mm)	
101	Reference point 1 Y(mm)	
102	Reference point 1 Z(mm)	
103	Reference point 1 A(mm)	
104	Reference point 2 X(mm)	
105	Reference point 2 Y(mm)	

106	Reference point 2 Z(mm)	
107	Reference point 2 A(mm)	
108	Reference point 3 X(mm)	
109	Reference point 3 Y(mm)	
110	Reference point 3 Z(mm)	
111	Reference point 3 A(mm)	
112	Reference point 4 X(mm)	
113	Reference point 4 Y(mm)	
114	Reference point 4 Z(mm)	
115	Reference point 4 A(mm)	

Explanation about Tool Parameter:

1,Mode of Setup Radius C Compensation (0:A type, 1:B type)

2,Mode of Cancel Radius C Compensation (0:A type,1:B type)

They are mode of setup/cancel radius C compensation, details at chapter3.26.

32,Filtering for Position Signal or WAT Signal

It is set filtering time for position signal of electric turret or WAT signal on special turret.

Time	Position Signal	WAT signal
2ms	+256	+2048
4ms	+512	+4096
8ms	+1024	+8192

Note: More details about tool parameters,please take Chapter5 as reference.

2.5 Other Parameter

P	Other Parameter	Ex-Factory
1	Type of Handwheel(0:Panel, 1:Handhold)	0
2	Type of Chuck	0
3	Using Interface Switch on Panel(0: No, 1:Yes)	1
4	Lubricate Automatically (0:Yes, 1:No)	0
5	Time of Lubrication (10ms)	300
6	Interval of Lubricate Automatically(s)	1800
7	Detection for Door Switch(0:No, 1:Yes)	0
8	Type of Door Switch(0:NO type, 1:NC type)	0
9	Special Bit Parameter	1000010001000101
10	Counting Workpieces Automatically(0:No, 1:Yes)	1
11	Increment of shift block	1
12	System Inner Parameter	9
13	Interlock between Rotation_SP & Chuck(0:No, 1:Yes)	1
13-1	Interlock between Rotation_SP & Tailstock(0:No, 1:Yes)	1
14	Coolant key is valid on Auto(0:No, 1:Yes)	1
15	Detect Position of Chuck(M10/M11)(0:No, 1:Yes)	0
16	Detect Position of Tailstock(M79/M78)(0:No, 1:Yes)	0
17	Type of Driver Alarm(ALM)(0:NO type, 1:NC type)	0
18	Type of Spindle Alarm(ALM1)(0:NO type, 1:NC type)	0
19	Type of Machine Alarm(ALM2)(0:NO type, 1:NC type)	0
20	Control Mode of Chuck(0:Single, 1:Double)	0
21	Control Mode of Tailstock(0:Single, 1:Double)	0
22	External Switch for Chuck(0:No, 1:Yes)	0
23	External Switch for Tailstock(0:No, 1:Yes)	0
24	Time of Chuck(s)	0.00
24-1	M10 Long Signal(0:No, 1:Yes)	0
24-2	M71 Long Signal(0:No, 1:Yes)	0
24-3	Chuck M10 or M11 boot [0:M10, 1:M11]	0
25	Time of Tailstock(s)	0.00
25-1	M79 Long Signal(0:No, 1:Yes)	0
25-2	M73 Long Signal(0:No, 1:Yes)	0
26	Type of Emergency Stop1(0:NO type, 1:NC type)	0
27	Type of Emergency Stop2(0:NO type, 1:NC type)	0
28	Run/Pause Output(0:No, 1:Yes)	0
29	Alarm Output(0:No, 1:Yes)	0
30	Language(1:Chinese, 0:English)	0
31	Use Inner PMC(0:No, 1:Yes)	1
32	Use High-Speed Inner PMC(0:No, 1:Yes)	1
33	HY as Running Key(0:No, 1:Yes)	0
34	HA as Halt Key(0:No, 1:Yes)	0
35	Soft-Limit is Valid when no homing(0:No, 1:Yes)	1
36	Time(Year-Month-Day-Hour-Minute)	
37	Rate of RS232	6
38	Latched for Rapid Key(8:Yes)	1
39	System Inner Parameter	1

40	System Inner Parameter	-88
41	Backup Current Parameters	
42	Recovery Backup Parameters	
120	Direction of Manual Feeding Key	1
200	X_Feedback Allow Error During Moving (Pulse) [>1]	1
201	Y_Feedback Allow Error During Moving (Pulse) [>1]	1
202	Z_Feedback Allow Error During Moving (Pulse) [>1]	1
203	A_Feedback Allow Error During Moving (Pulse) [>1]	1
204	B_Feedback Allow Error During Moving (Pulse) [>1]	1
205	X_Feedback Allow Error When Stopping (Pulse) [>1]	1
206	Y_Feedback Allow Error When Stopping (Pulse) [>1]	1
207	Z_Feedback Allow Error When Stopping (Pulse) [>1]	1
208	A_Feedback Allow Error When Stopping (Pulse) [>1]	1
209	B_Feedback Allow Error When Stopping (Pulse) [>1]	1
210	Numerator of X_Electron Gear for Position Feedback	1
211	Numerator of Y_Electron Gear for Position Feedback	1
212	Numerator of Z_Electron Gear for Position Feedback	1
213	Numerator of A_Electron Gear for Position Feedback	1
214	Numerator of B_Electron Gear for Position Feedback	1
215	Denominator of X_Electron Gear for Position Feedback	1
216	Denominator of Y_Electron Gear for Position Feedback	1
217	Denominator of Z_Electron Gear for Position Feedback	1
218	Denominator of A_Electron Gear for Position Feedback	1
219	Denominator of B_Electron Gear for Position Feedback	1
300	Feeding Axes with Absolute Servo(0:No, 1:Yes)	00000001
301	Multi-turns Lower 16bits data Address of Absolute Encoder	92
302	Single-turn Higher 16bits data Address of Absolute Encoder	91
303	Single-turn Lower 16bits data Address of Absolute Encoder	90
304	Pulses of One-Revolution in X-Axis(Absolute)	131072
305	Pulses of One-Revolution in Y-Axis(Absolute)	131072
306	Pulses of One-Revolution in Z-Axis(Absolute)	131072
307	Pulses of One-Revolution in A-Axis(Absolute)	131072
308	Pulses of One-Revolution in B-Axis(Absolute)	131072
309	Pitch of ball screw in X-Axis(nm)(Absolute)	10000000
310	Pitch of ball screw in Y-Axis(nm)(Absolute)	10000000
311	Pitch of ball screw in Z-Axis(nm)(Absolute)	10000000
312	Pitch of ball screw in A-Axis(nm)(Absolute)	10000000
313	Pitch of ball screw in B-Axis(nm)(Absolute)	10000000
314	Offset of Multi-turns in X-axis Absolute Encoder	1
315	Offset of Multi-turns in Y-axis Absolute Encoder	1
316	Offset of Multi-turns in Z-axis Absolute Encoder	1
317	Offset of Multi-turns in A-axis Absolute Encoder	1
318	Offset of Multi-turns in B-axis Absolute Encoder	1
319	16bits Data Address for Current/Speed/Torque of Absolute Driver	284
320	Ratio of Gearbox in X-Axis(Absolute)	1
321	Ratio of Gearbox in Y-Axis(Absolute)	1
322	Ratio of Gearbox in Z-Axis(Absolute)	1
323	Ratio of Gearbox in A-Axis(Absolute)	1

324	Ratio of Gearbox in B-Axis(Absolute)	1
500	Series Number of Ex-Factory	0.00
501	Shift Color Display of Screen(1:No, 8:Yes)	1
601	Define Parameters for Step	
602	Define Parameters for Servo	Servo
900	Display User-define Dialog Box[1:No, 4:Yes]	4
901	Homing Sequence of Axis(5bits)	1
910	High-Speed Input of M18/M22/M24/M28 for G31/G311	0
911	Using M18_Teach-in, M28_Record(0:No, 1:Yes)	0
912	"Reset"key reset Outputs(0:No, 1:Yes)	1

Note:P12&P39&P40 are System Inner Parameter, cannot be altered.

Explanation about Axis Parameter:

1,Type of Handwheel(0:Panel, 1:Handhold)

It sets the type of handwheel,0:Handwheel in operational panel, 1:Handwheel in handhold box.

Note: when the parameter is 1 (P1=1), CN11 is connected to handhold box; & can't use rate of spindle&feeding axis,only off/X/Y/Z/A &*1/*10/*100(also P1&P2=0 in Axis parameter).

2,Type of Chuck

It set type of chuck, 0: Inside Chuck(M10: chuck clamp to center); 1:Outside Chuck(M10: chuck clamp to external).

3,Using Interface Switch on Panel(0: No, 1:Yes,)

It sets whether system use interface switch on operational panel. 0:No, don't use interface switch; 1:Yes,use interface switch.

4,Lubricate Automatically (0:Yes, 1:No)

It sets whether system use lubricate automatically. 0:Yes, lubricate automatically is valid, 1:No use lubricate automatically.

Attention:Lubricate automatically according to time of running program.

5,Time of Lubrication (10ms)

It sets the time of lubricate automatically , also time of outputting M32, PIN9_CN3 Plug. Unit:0.01s.

6,Interval of Lubricate Automatically(s)

It is the interval that lubricate every time,also the interval that twice M32 is valid.

7,Detection for Door Switch(0:No, 1:Yes)

It sets whether system detect the signal of protective-door. 0:No detect, 1:Yes.

Attention:1. Input point for door switch: M12, PIN11_CN10 plug.

2. After set P7=1,system can work in Manual,and stop processing in Auto.

3. Pin for detect Chuck_clamp&Door-switch are M12, only one usage is valid.

8,Type of Door Switch(0:NO type, 1:NC type)

It is type of Door-switch. 0:NO type(normal open),1:NC type(normal close).

9,Special Bit Parameter

It is bit parameter,each one bit have different functions,detailes as following:

Bit	D15	D14	D13	D12	D11	D10	D9	D8	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1	0	1	0	0	0	1	0	1

D0: Null; default value is 1,which cannot be altered.

D1: 1:Clear Part Number after reboot system; 0:Keep Part number.

D2: 1:Indent automatically between characters when edit; 0:No blank;

D3: Null; default value is 0,which cannot be altered.

D4: Null; default value is 0,which cannot be altered.

D5: 1:Don't stop Rotation_SP & Coolant when pressing "Reset" key;

D6: 1:Each axis run with itself speed when G00; 0: linkage movement;

D7: 1:Don't call related tool compensation when tool change manually; 0: Call related tool compensation automatically; default is 0.

D8: 1:Save status of Chuck(M10/M11) when power off;Recovery original status when booting system; 0: System output M10 automatically when booting.

D9: Select Mode of tool set & input mode of Redeem;

D10: 1:Auto Sequence for block when programming;

D11: 1:Analog of 1st spindle outputs to both +10V_CN3&CN10;

D12: 1:Shield function of "Skip" ,also "/" in the front of blocks is invalid;

D13: 1:Shield function of "Return" key on operational panel;

D14: 1:Shield function of "Start" key on operational panel;

D15: 1:Value of Redeem displays with Increment type; 0: Value of Redeem displays with absolute type;

Attention:some bits of this bit parameter cannot be altered , otherwise it maybe system will work abnormal.

10,Counting Workpieces Automatically(0:No, 1:Yes)

It set whether system counting number of workpiece automatically, 0:No counting workpieces automatically; 1:Yes,counting automatically.

11,Increment of shift block

It sets the increment of block when change lines.

12,System Inner Parameter

- It is system inner parameter,which cannot be altered.

13,Interlock between Rotation_SP & Chuck(0:No, 1:Yes)

It sets interlock between rotation of spindle and Chuck(M10). 0:No interlock, rotation of spindle isn't related to Chuck; 1:Yes,only when chuck is clamping,and then spindle can rotate; when spindle is rotating, Chuck cannot unclamping.

13-1,Interlock between Rotation_SP & Tailstock (0:No, 1:Yes)

It sets interlock between rotation of spindle and tailstock(M79). 0:No interlock, rotation of

spindle isn't related to Tailstock; 1:Yes,only when tailstock forward, and then spindle can rotate;

14,Coolant key is valid on Auto(0:No, 1:Yes)

It sets if Press "Coolant"key is valid on Auto. 0:No,"Coolant" key doesn't work on Auto; 1:Yes, "Coolant" key also works in condition of Auto.

15,Detect Position of Chuck(M10/M11)(0:No, 1:Yes)

It sets if detect position of chuck. 0:No detect; 1:Yes,detect.

If P15=1, M12,PIN11_CN10 Plug, position input for Chuck(Clamp/M10); M14 , PIN24_CN10 plug, position input for Chuck(Loose/M11)

Attention: It is same pin(M12) of check of Chuck-clamp&Door-switch, only one usage is valid.If check chuck clamp if is in position,also cannot be used as check of Door-switch.

It is same pin(M14) of check of Chuck-loose&Tailstcok-switch,only one usage is valid. If check chuck-loose if is in position,also cannot be used as Tailstock control switch.

16,Detect Position of Tailstock(M79/M78)(0:No, 1:Yes)

It sets if detect position of tailstock. 0:No detect; 1:Yes, detect;

If P16=1, M18,PIN10_CN10 Plug, position input for Tailstock(Forward/M79; M28,PIN23_CN10 Plug, position input for Tailstock(Backward/M78).

17,Type of Driver Alarm(ALM)(0:NO type, 1:NC type)

It sets the type of driver alarm. ALM,PIN12_CN5 plug, 0:NO type; 1:NC type.

18,Type of Spindle Alarm(ALM1)(0:NO type, 1:NC type)

It sets the type of spindle alarm. ALM1,PIN5_CN3 plug,0:NO type; 1:NC type.

19,Type of Machine Alarm(ALM2)(0:NO type, 1:NC type)

It sets the type of machine_tool alarm. Input point is ALM2, PIN2_CN10 plug. 0:NO type; 1:NC type.

20,Control Mode of Chuck(0:Single, 1:Double)

It sets the control mode of chuck, 0:Single control signal for Chuck; 1:Double control signal for Chuck.

P20=0, one output point for Chuck, M10:clamp chuck, M11: unclamp chuck;

P20=1, two output points for Chuck, M10: output M10(PIN21_CN3 Plug) to clamp chuck; M11: output M71(PIN9_CN10 Plug) to unclamp chuck.

21,Control Mode of Thumbstall(0:Single, 1:Double)

It sets the control mode of tailstock, 0:Single control signal for thumbstall, also tailstock; 1:Double control signal for Thumbstall.

P21=0, one output point for thumbstall, M79:thumbstall forward, M78, also M79 is invalid: thumbstall backward;

P21=1, two output points for thumbstall, M79: output M79(PIN22_CN3 Plug) to forward thumbstall; M78: output M73(PIN22_CN10 Plug) to backward thumbstall.

22,External Switch for Chuck(0:No, 1:Yes)

It sets if there is external switch for control chuck. 0:No,without switch for chuck;1:Yes,with external switch for chuck. Input point is M16,PIN12_CN10 Plug.

Note: It is reciprocating signal. one is valid,clamp chuck; another is invalid,loose chuck.

23,External Switch for Tailstock(0:No, 1:Yes)

It sets if there is external switch for control tailstock,0:No,without switch for tailstock; 1:Yes,with switch for tailstock. Input point is M14,PIN24_CN10 Plug.

Note:Reciprocating signal.one is valid,tailstock forward;anther is invalid,tailstock backward.

24,Time of Chuck(s)

It sets holding time of output M10/M11 for chuck. Unit:second. 0 means M10/M11 are long signal, also always output M10/M11 is valid.

24-1,M10 Long Signal(0:No, 1:Yes)

It sets the control mode of M10, 0 means short signal, holding time of output M10 can be set by parameter, 1 means long signal, M10 always valid

24-2,M71 Long Signal(0:No, 1:Yes)

It sets the control mode of M71, 0 means short signal, holding time of output M71 can be set by parameter, 1 means long signal, M71(M11) always valid.

24-3,Chuck M10 or M11 boot [0:M10, 1:M11]

It sets CNC system output M10 or M11 when booting system, 0:Output M10, 1:Output M11.

Note: If double outputs for chuck, M11,also output M71, PIN9_CN10 Plug.

25,Time of Tailstock(s)

It sets holding time of output M79/M78 for tailstock. Unit:second. 0 means M79/M78 are long signal, also always output M79/M78 is valid.

25-1,M79 Long Signal(0:NO, 1:Yes)

It sets the control mode of M79, 0 means short signal, holding time of output M79 can be set by parameter, 1 means long signal, M79 always valid

25-2,M73 Long Signal(0:NO, 1:Yes)

It sets the control mode of M73(M78), 0 means short signal, holding time of output M73 can be set by parameter, 1 means long signal, M73(M78) always valid.

Note:If double outputs, M78, also output M73, PIN22_CN10 Plug.

26,Type of Emergency Stop1(0:NO type, 1:NC type)

It set thee type of switch for 1st Emergency Stop, which is at panel. 0: NO type switch; 1:NC type switch for 1st emergency stop.

27,Type of Emergency Stop2(0:NO type, 1:NC type)

It set thee type of switch for 2nd Emergency Stop, which is at panel. 0: NO type switch; 1:NC type switch for 2nd emergency stop.Input is PIN5_CN11 Plug.

28,Run/Pause Output(0:No, 1:Yes)

It sets if output the condition of Running/Pause. 0:No, don't output condition of Run/Pause; 1:Yes, output the condition of Run/Pause. And M69, PIN21_CN10 plug , output Running condition; M65,PIN20_CN10 plug, output Pause condition.

Note:These signals can be used to indicator for show condition of machine.

29,Alarm Output(0:No, 1:Yes)

It sets if output the condition of Alarm. 0:No, don't output condition of Alarm; 1:Yes, output the condition of Alarm, Output point is M67,PIN8_CN10 Plug.

Note: The signals can be used as machine-protection or show condition of machine.

30,Language(1:Chinese, 0:English)

It sets the language of system. 1: Set language to Chinese ; 0: set to English.

31,Use Inner PMC(0:No, 1:Yes)

It sets if use inner PMC function; 0:No, no use; 1:Yes, use.

Warning:It is usually used for adjusting parameters. system must use inner IO PMC when actual use,also P31=1. Otherwise system will works abnormally.

32,Use High-Speed Inner PMC(0:No, 1:Yes)

It sets if use high-speed inner PMC for IOs. 0:No,don't use PMC; 1: Yes,use High-Speed PMC.

Warning:It is usually used for adjusting parameters. system must use High-Speed PMC when actual use,also P32=1. Otherwise system will works abnormally.

33,HY as Running Key(0:No, 1:Yes)

It sets if make HY input point, PIN9_CN11 plug as external key for RUN signal. 0:No, HY don't as RUN input signal; 1:Yes,HY as RUN signal.

Attention: Because HY signal maybe as Y(C)-axis selection signal, so when P33=1,then P1 in Axis&Other parameter only set as 0.

34,HA as Halt Key(0:No, 1:Yes)

It sets if make HA input point, PIN10_CN11 plug as external key for STOP signal. 0:No, HY don't as Halt input signal; 1:Yes,HA as Halt signal.

Attention:Because HA signal maybe as A-axis selection signal, so when P33=1,then P1 in Axis&Other parameter only set as 0.

35,Soft-Limit is Valid when no homing(0:No, 1:Yes)

It sets if soft-limit is valid when not homing. 1:Yes,valid, 0:No,invalid.

Attention:the set of this parameter is related to operation habits.

36,Time(Year-Month-Day-Hour-Minute)

It sets time and date of system. After set well,system will take this setting time as basic,according to inner timer count time and shows in displayer.

Example:13:33, 16th, March, 2017; set P36=2017-03-16-13-33, & Enter.

37,Rate of RS232

It sets rate of communication with RS232. Different value corresponding to different rate:[0=7200;1=9600;2=14400;3=19200;4=38400;5=57600; 6=115200].

Attention:The Rate of both CNC & PC must keep same.

38,Latched for Rapid Key(8:Yes)

It sets if latched for "Rapid" key on panel. 8:Yes. Reciprocating control.

41,Backup Current Parameters

It is for backup current parameters as ex-factory set. It is used for backup parameters after debugging is finished well, easy to maintain.

Attention: select this parameter,press "Enter" key twice,finish backups.

42,Recovery Backup Parameters.

It is for recovery current parameters to ex-factory set. It is normally used for recovery to ex-factory set when parameters set wrong.

Attention: after finish this operation,last parameter will be occupied.

120,Direction of Manual Feeding Key

It sets feeding direction of manual feeding key of each axis on panel.

Value	Function
+4	Direction of Z_Manual Feeding is opposite;
+8	Direction of C(Y)_Manual Feeding is opposite;
+16	Direction of X_Manual Feeding is opposite;
+32	Direction of A_Manual Feeding is opposite;

200,X_Feedback Allow Error During Moving (Pulse) [>1]
201,Y_Feedback Allow Error During Moving (Pulse) [>1]
202,Z_Feedback Allow Error During Moving (Pulse) [>1]
203,A_Feedback Allow Error During Moving (Pulse) [>1]

If follow error bigger than this parameter,system will alarm.(the parameter is effective when it >1). #200/#201/#202/#203 stand for X/Y/Z/A-axis. Press "G"key in diagnosis screen to clear alarm and command position & feedback position after alarm.

205,X_Feedback Allow Error When Stopping (Pulse) [>1]
206,Y_Feedback Allow Error When Stopping (Pulse) [>1]
207,Z_Feedback Allow Error When Stopping (Pulse) [>1]
208,A_Feedback Allow Error When Stopping (Pulse) [>1]

If follow error bigger than this parameter when X-axis stop,system will alarm.(the parameter is effective when it >1). #205/#206/#207/#208 stand for X/Y/Z/A-axis. Press "G"key in diagnosis

screen to clear alarm and command position & feedback position after alarm.

210, Numerator of X_Electron Gear for Position Feedback
211,Numerator of Y_Electron Gear for Position Feedback
212,Numerator of Z_Electron Gear for Position Feedback
213,Numerator of A_Electron Gear for Position Feedback

The parameter is numerator of X-axis electron gear,and also input screw lead and encoder lines,for example,when screw leas is 6mm,encoder's resolution is 2500PPR,input : L6000M2500. #210/#211/#212/#213/#214 stand for X/Y/Z/A-axis.

Special attention:

1)when input LM***,it needs to set numerator and denominator of electron gear of each axis.***

2)when series port read data of absolutely encoder,it must input in this way.

215,Denominator of X_Electron Gear for Position Feedback
216,Denominator of Y_Electron Gear for Position Feedback
217,Denominator of Z_Electron Gear for Position Feedback
218,Denominator of A_Electron Gear for Position Feedback

The parameter is denominator of X-axis electron gear,and also input screw lead and encoder lines,for example,when screw leas is 6mm,encoder's resolution is 2500PPR,input : L6000M2500. #215/#216/#217/#218 stand for X/Y/Z/A-axis.

Special attention:

1)when input LM***,it needs to set numerator and denominator of electron gear of each axis.***

2)when series port read data of absolutely encoder,it must input in this way.

300,Feeding Axes with Absolute Servo(0:No, 1:Yes)

Bit	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1
Func	C	B	A	Z	Y	X	-	-

It is bit parameter, and sets if each feeding axes is with absolute servo or not. 0 means without absolute servo; 1 means yes, with absolute servo.

301,Multi-turns Lower 16bits data Address of Absolute Encoder

It is multi-turns lower 16bits data address of absolute encoder, Example: set P301=92.

302,Single-turn Higher 16bits data Address of Absolute Encoder

It is single-turn higher 16bits data address of absolute encoder, Example: set P302=91.

303,Single-turn Lower 16bits data Address of Absolute Encoder

It is single-turn lower 16bits data address of absolute encoder, Example: set P303=90.

Special Attention: System is connected to absolute type servo driver with RS485 by CN13 port, communication protocol is as follow:

1) Driver's Station Address of RS485:1 -X axis; 2 -Y axis; 3 -Z axis; 4 -A axis ; 5 - B axis.

2) Baud Rate of RS485 is 19200;Data is 8 bits;Stop bit is 1 bit; Format is RTU; Parity bit is odd. {1start+8data+1stop+odd}

304,Pulses of One-Revolution in X-Axis(Absolute)
305,Pulses of One-Revolution in Y-Axis(Absolute)
306,Pulses of One-Revolution in Z-Axis(Absolute)
307,Pulses of One-Revolution in A-Axis(Absolute)
308,Pulses of One-Revolution in B-Axis(Absolute)

It sets pulses of one-revolution in X-axis, which is configured with absolute encoder. The pulses of 17bits absolute encoder is 131072. #304/#305#306/#307/#308 stand for X/Y/Z/A/B-axis.

309,Pitch of ball screw in X-Axis(nm)(Absolute)
310,Pitch of ball screw in Y-Axis(nm)(Absolute)
311,Pitch of ball screw in Z-Axis(nm)(Absolute)
312,Pitch of ball screw in A-Axis(nm)(Absolute)
313,Pitch of ball screw in B-Axis(nm)(Absolute)

It sets pitch of ball screw in X-axis, which is configured with absolute encoder ,also distance per revolution of ball screw. Minus means count's direction is negative. When read data of absolute encoder is too much differ to before, set to opposite number before.

Example: Pitch of ball screw in X-axis is 4mm, so P309=4000000 or -4000000. #309/#310 /#311/#312/#313 stands for X/Y/Z/A/B-axis.

314,Offset of Multi-turns in X-axis Absolute Encoder
315,Offset of Multi-turns in Y-axis Absolute Encoder
316,Offset of Multi-turns in Z-axis Absolute Encoder
317,Offset of Multi-turns in A-axis Absolute Encoder
318,Offset of Multi-turns in B-axis Absolute Encoder

It sets multi-tuns offset of X-axis,in order to slove when the values of multi-turns or coordinate value is too big ,and overflow etc. problems. Input 'E' clear the value to 0 . #314/#315/#316 /#317/#318 stand for X/Y/Z/A/B-axis.

319,16bits Data Address for Current/Speed/Torque of Absolute Driver

It sets address of 16bits data for current, speed & torque in absolute driver. #284:current; #283:Speed; #435: Torque.

320,Ratio of Gearbox in X-Axis(Absolute)
321,Ratio of Gearbox in Y-Axis(Absolute)
322,Ratio of Gearbox in Z-Axis(Absolute)
323,Ratio of Gearbox in A-Axis(Absolute)
324,Ratio of Gearbox in B-Axis(Absolute)

It is ratio of gearbox in X-axis,which is configured with absolute encoder.Eg.: X:Pitch is 4mm, ratio is 3:1, so P309=4000000,P320=3. #320/#321/#322/#323/#324 stands for X/Y/Z/A/B axis.

500,Series Number of Ex-Factory

It sets series number of ex-factory.

501,Shift Color Display of Screen(1:No, 8:Yes)

It sets if shift color display of screen, 1: No shift; 8:Yes,shift to black color.

601,Define Parameters for Step

It sets current parameters to ex-factory set for step system when machine tool is configured with stepper motor&driver.The operation is done before debugging.

602,Define Parameters for Servo

It sets current parameters to ex-factory set for servo system when machine tool is configured with servo motor&driver.The operation is done before debugging.

900,Display User-defined Dialog Box[1:No, 4:Some, 8:All]

It sets if display user-define dialog box. 1: No display; 4:Yes,display some.

901,Homing Sequence of Axis(5bits)

It sets homing sequence of each axis.Value is 5bits.D0 bit is 0. 1:X, 2:C(Y), 3:Z, 4:A. Eg.: P901=31240, Homing sequence is Z->X->Y->A.

910,High-Speed Input of M18/M22/M24/M28 for G31/G311(0:No, 1:Yes)

It sets if inputs of M18/M22/M24/M28 are high-speed input for G31/G311 command; 0:No, don't as input for G31/G311; 1:Yes.

911,Use M18_Teach-in, M28_Record(0:No, 1:Yes)

It set if use M18 as Teach-in function, M28 as Set function of Teach-in.

912,"Reset"key reset Outputs(0:No, 1:Yes)

It sets if "Reset"key reset output points. 0:No reset outputs; 1: Yes, reset.

SZGH-CNC1000MDc CNC system can be configured with Delta absolute servo driver, related parameters set as following:

Parameters set on CNC system	Parameters set on Delta Driver
P301=102 (0x66 of P0_51 in delta)	P3_00, Station No.; 1/2/3/4/5 is corresponding to X/Y/Z/A/B
P302=104 (0x68 of P0_552 in delta)	P3_01=2
P303=-98 (0x62 of P0_49 in delta)	P3_02=8
P304 = 1280000	P3_03=0
P305 = 1280000	P3_04=0
P306 = 1280000	P3_05=1
P307 = 1280000	P3_07=0
P308 = 1280000	P2_69=1
	P2_70=2

2.6 Workpiece Coordinate Parameter

CNC system supports multiple coordinate system function, also 6 workpiece coordinate system(G54-G59), plus 10 workpiece coordinate system(G54.1-G54.10) and a machine coordinate system G53. A machining program can set a workpiece coordinate system can also be set up multiple workpiece coordinate system, the workpiece coordinate system can be changed to move its origin. That is the value of the parameter in the coordinates of its own coordinate origin (zero) coordinate value in the machine coordinate system.

In Lathe System,normally user only need one coordinate system(G53 coordinate system), also Machine Coordinate System.

G54 to G59 can be set with 6 workpiece coordinate systems, the coordinate system settings interface can be modified 6 origin of the workpiece coordinate system coordinate value in the machine coordinate system.

P	Coordinate Parameter	Ex-Value
1-0	Current Workpiece Coordinate Set [G54-G59]	54
1-1	X_Workpiece Coordinate (G54-G59)	0.000
1-2	Y_Workpiece Coordinate (G54-G59)	0.000
1-3	Z_Workpiece Coordinate (G54-G59)	0.000
1-4	A_Workpiece Coordinate (G54-G59)	0.000
2-0	Current Workpiece Coordinate Set [G54.1-G54.10]	1
2-1	X_Workpiece Coordinate (G54.1-G54.10)	0.000
2-2	Y_Workpiece Coordinate (G54.1-G54.10)	0.000
2-3	Z_Workpiece Coordinate (G54.1-G54.10)	0.000
2-4	A_Workpiece Coordinate (G54.1-G54.10)	0.000
1	X_Workpiece Coordinate of G54	0.000
2	Y_Workpiece Coordinate of G54	0.000
3	Z_Workpiece Coordinate of G54	0.000
4	A_Workpiece Coordinate of G54	0.000
6	X_Workpiece Coordinate of G55	0.000
7	Y_Workpiece Coordinate of G55	0.000
8	Z_Workpiece Coordinate of G55	0.000
9	A_Workpiece Coordinate of G55	0.000
11	X_Workpiece Coordinate of G56	0.000
12	Y_Workpiece Coordinate of G56	0.000
13	Z_Workpiece Coordinate of G56	0.000
14	A_Workpiece Coordinate of G56	0.000
16	X_Workpiece Coordinate of G57	0.000
17	Y_Workpiece Coordinate of G57	0.000
18	Z_Workpiece Coordinate of G57	0.000
19	A_Workpiece Coordinate of G57	0.000
21	X_Workpiece Coordinate of G58	0.000
22	Y_Workpiece Coordinate of G58	0.000
23	Z_Workpiece Coordinate of G58	0.000
24	A_Workpiece Coordinate of G58	0.000
26	X_Workpiece Coordinate of G59	0.000
27	Y_Workpiece Coordinate of G59	0.000
28	Z_Workpiece Coordinate of G59	0.000
29	A_Workpiece Coordinate of G59	0.000
.....		

Note:

1. When CNC controller is with related axes,which has related functions for feeding axes.

2. Input "E" to clear coordinate value.

3. Value Set for parameter of workpiece coordinate system is with increment type.

3. Each alone G54.1-G54.10 workpiece coordinate set just don't display on user manual.

Explanation about Workpiece Coordinate System:

1-0,Current Workpiece Coordinate Set [G54-G59]

It is for select current workpiece coordinate from G54 to G59.

1-1 X_Workpiece Coordinate (G54-G59)
1-2 Y_Workpiece Coordinate (G54-G59)
1-3 Z_Workpiece Coordinate (G54-G59)
1-4 A_Workpiece Coordinate (G54-G59)

It sets value of related axis on workpiece coordinate system, which is set by P1-0. The value is set with Increment type.

2-0 Current Workpiece Coordinate Set [G54.1-G54.10]

It is for select current workpiece coordinate from G54.1 to G54.10

2-1 X_Workpiece Coordinate (G54.1-G54.10)
2-2 Y_Workpiece Coordinate (G54.1-G54.10)
2-3 Z_Workpiece Coordinate (G54.1-G54.10)
2-4 A_Workpiece Coordinate (G54.1-G54.10)

It sets value of related axis on workpiece coordinate system, which is set by P2-0. The value is set with Increment type.

2.6.1 How to set up the workpiece coordinate system?

We set up the workpiece coordinate in the condition of Manual,the steps are following:

a).Press "MDI"key,select corresponding workpiece coordinate system(G54-G59), **Example,select G55 coordinate,input G55 ,Press 'Enter', 'Start',selected G54 Coordinate.**

b).Move machine to suitable position that easy to measure in manual,and measured the related coordinate value between this point (zero point in the workpiece)to Home of G53 coordinate system(also machine coordinate system).

c).Press"Setup",press "X"key and 'Enter','insert the measured value',and 'Enter'.

d).Press "Setup",press "Z"key and 'Enter','insert the measured value',and 'Enter'

e).Press"Setup",press "Y"key and 'Enter','insert the measured value',and 'Enter'.

f).Press "Setup",press "A"key and 'Enter','insert the measured value',and 'Enter'

Done well now.Enter different workpiece coordinate system,it will show the corresonding value,which also is offset value between workpiece coordinate system and machine coordinate system(G53).

2.6.2 How to adjust the offset value after set well?

If set up workpiece coordinate system well,when it needs to adjust the offset value,it could be set by enter the workpiece coordinate parameter,steps is as follow:

In the coordinate parameter screen,selected the parameter,press "Enter",and pop up dialog,input the offset value(also Increments,example:offset 10mm in negative direction,also input -10),press "Enter".It is okay.

Explanation:1.when the parameter is altered well,the coordinate main screen will refresh the corresponding coordinate value soon.

2.brackets in these parameters,it means the sum ,which is offset or adjust every time.It is suitable to look for the offset every time.

2.7 Password

The password is order to avoid modified accidentally and ensure the system work in normal condition.The system adopt three permissions, “CNC Factory”, “Machine Factory” and “User”.

The original condition is “CNC factory” is set, “Machine factory” and “User” isn’t set.

After set new password(set new password ,it need original password),please remember the new password ,and the original password wasn’t work.

Attention:the password must be 6 bit data,the data could be number and letter.

password setting include:

1,Is enable CNC Co.’s password ?

It is for inner parameter,it couldn’t be operated.

2,Is enable Machine Co.’s password ?

Display and set the parameter that is related to machine’s configuration.

3,Is enable User’s password ?

It is for whether display and set the parameter that is related to processing.

4,Modify CNC Co.’s password:

5,Modify Machine Co.’s password:

6,Modify User’s password:

7,curry word time: (days)

Version of Operational Software.

Ex-factory Time

2.8 Redeem

Press “Redeem” key to enter interface of redeem in any condition.

Remark	Function
F1-Radius	Press “F1” key to enter Radius Compensation Interface
F2-Length	Press “F2” key to enter Length Compensation Interface
F3-ACLEA	Press “F3” key to clear all compensation value.
F4-CLEAR	Press “F4” key to clear current compensation value.
F5-SetTool	Press “F5” key to set tool {same to Setup key on panel}
F6-ToolPoit	Press “F6” key to enter list of ToolPosit
F7-Set	Press “F7” key to set total tool number
F8-CANCEL	Press “F8” key to return back main interface

2.8.1 Radius Compensation

Press “F1” to entering radius compensation interface on Redeem.

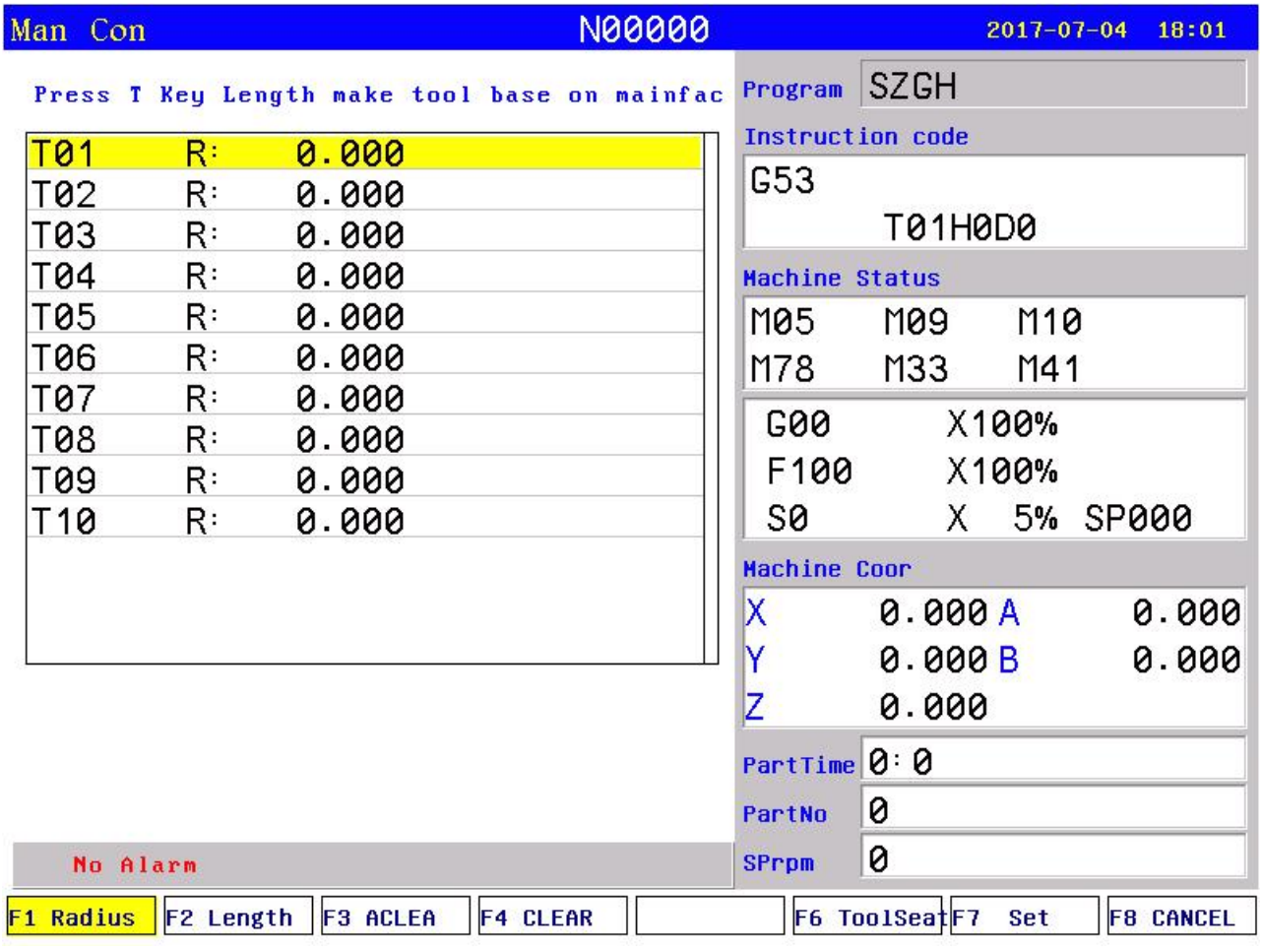


Fig2.8.1 Radius Compensation Interface

Setting Steps: Press “↑ ↓” key to move cursor to related tool and press “Enter” key to pop-up a dialog box “Input T# tool radius compensate R:”, input radius value of corresponding tool, press “Enter” at last.

Note: Value input is with absolute type.

2.8.2 Length of redeem

Press “F2” to enter Tool Length offset interface on Redeem.

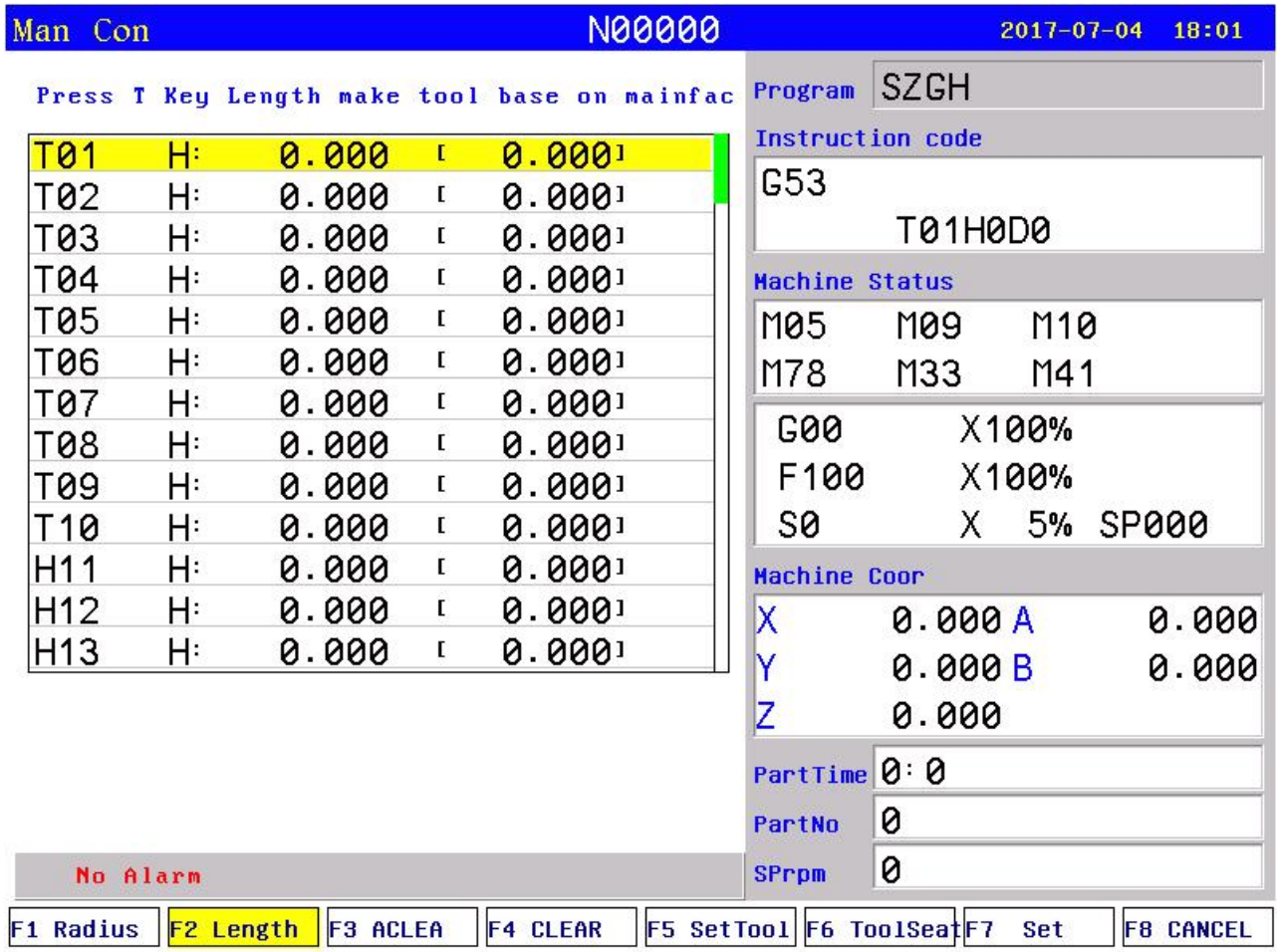


Fig2.8.2 Tool Length Offset Interface

Steps of modifying length compensation:

Press “↑ ↓” key to move cursor to the corresponding tool number and press “Enter” to pop-up a dialog box, import the modifying axis into the dialog box and import the modifying value(import 0.05 to plus 0.05, import -0.05 to reduce 0.05), press “Enter” to confirm. The system calculates current value of redeem after finishing setting.

Method of Automatic Tool Set

1)Move machine tool to a position where is easy to measure coordinate of tools

2)Press “↑ ↓” to move cursor to corresponding tool number

3)Press “F5” to pop-up a dialog box, “input axis name:[X,Y,Z,A,B]”

4)Press X/Y/Z/A/B key, “Input Coordinate Axis(mm): X/Y/Z/AB”, input coordinate value

4)Press “Enter” to confirm.compensation of corresponding axis is set well.

The system refresh current value of redeem after finishing setting automatically.

Method of initializing the length compensation value of tool:

Press “F3” or “F4” to initialize length compensation of all or current tool.

Note: Value input is with increment type.

2.8.3 Tool Sets List

Press “F6” to enter posit tool interface in redeem. The parameter is used to set type of tool sets when adopting radius compensation of tool.

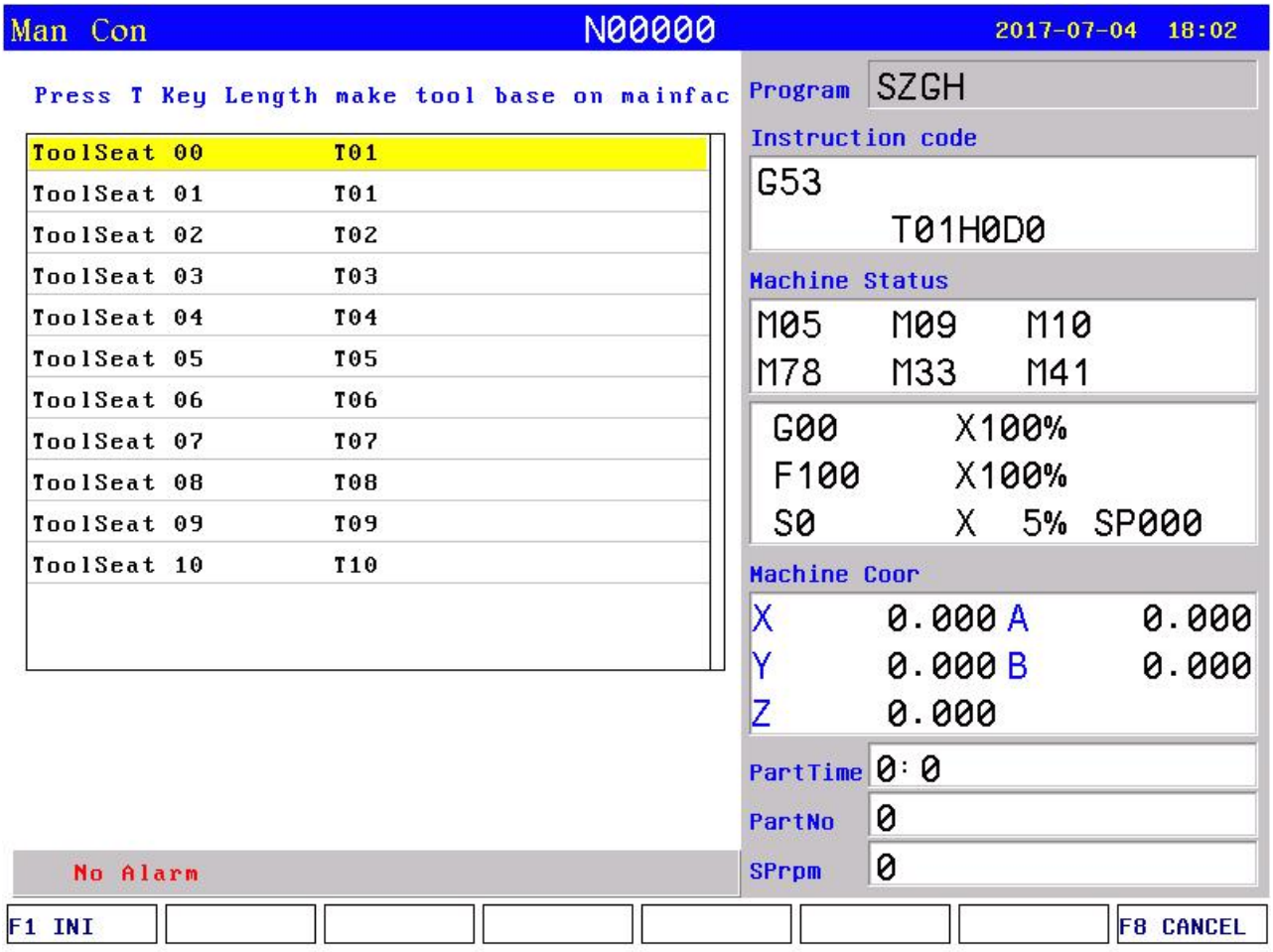


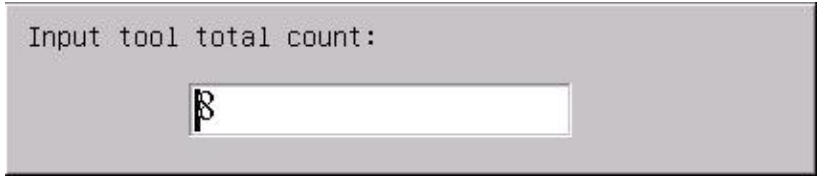
Fig2.8.3 Tool Posit Interface

Step of setting: Press “↑ ↓” to move cursor to corresponding tool number and press “Enter” to pop-up a dialog box, input the code of corresponding tool’s types and press “Enter” to confirm.

Press “F1” key to initialize all the kinds of tool point to 0.

2.8.4 Set quantity

Press “F7” key to pop-up a dialog box On Redeem interface to set total tools.



Including sum tools of electrical tools , linear tools and tool-post.

The CNC system supports 99 pieces of tools max.

2.9 Screw Compensation

Press “Parameter” twice to enter screw compensation interface.

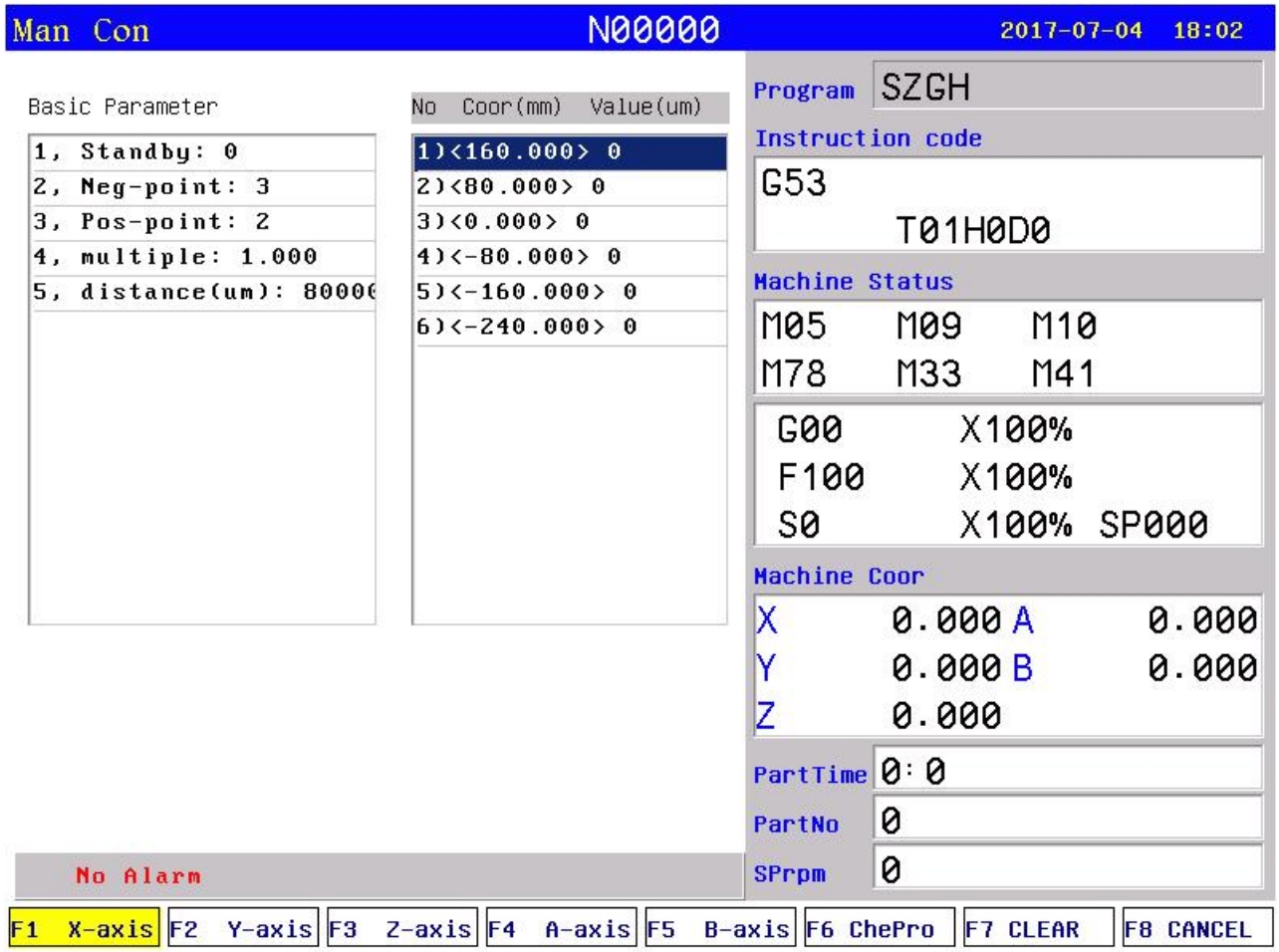


Fig2.9.1 Screw compensation interface

Screw compensation is used for Automatic compensating the error of screw pitch, which due to the error of screw pitch to affect accuracy of machine. The system adopts built-in screw compensation: Take machine’s home position, also datum point as the starting point when debugging, measured the error curve of screw, studied out the correctional curve according to the error curve, import the value of correctional curve into the correctional parameter and system is going to compensate according to the parameter in Automatic running.

Screw compensation by the axis as the unit to set storage, set X/Z/C/A/B axis separately, by pressing “F1” “F2” “F3” “F4” “F5” to switch; Every axis of screw compensation interface has tow areas(basic parameter and set the compensation), by pressing “→ ←”to move the cursor to realize.

Storage of screw compensation curve is with each axis, set screw compensation of X Y Z A B axis separately, by pressing “F1” “F2” “F3” “F4” “F5” to switch; Every axis of screw compensation interface has two areas(basic parameter and set the compensation), which switch is through pressing “→ ←”to move the cursor .

A) Basic parameter:

Press “↑ ↓” to select current basic parameter to set in basic parameter, press “Enter” to pop-up a dialog box to import the error compensation of every axis and import the basic information of

screw compensation.

Basic parameter of every axis' error compensation of screw pitch includes as follows:

1. Reserve.

2. Backward checking points.

It is set for points number of compensation in negative direction.

3. Forward checking points.

It is set for points number of compensation in positive direction.

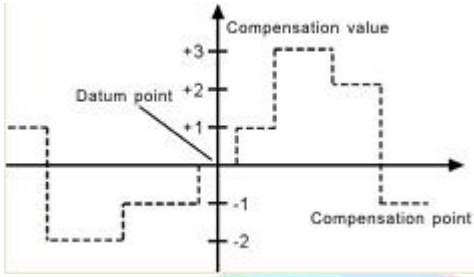
4. Multiple.

It is set for rate of compensation, also actual value=set value * multiple .

5. Distance (um).

It is set for the distance between two compensate points.

Note: Number of compensation points can be set freely, max points of each axis is 300.



B) Set compensation value (No. Coor(mm) Value(um)):

In the area of setting compensation value, it will shows the value of compensation and every axis' error compensation point of screw pitch. Press "↑ ↓ PgDn PgUp" to select current compensation point and press "Enter" to pop-up a dialog box to import the value of current compensation point.

Test program generation automatically

Automatic generate a program of laser interference device to check the screw compensation. Enter the screw compensation screen and set basic parameters well, press "B" key to check program" to detect program to pop-up a dialog box and press "Enter" to generate corresponding checking program of screw compensation.

System calculates the distance of compensation points automatically according to basic parameter. Distance is uniform, which could be set according to different axis, and user can set compensation value of each point (System requires input absolute value, relating to value of datum point).

Example1: Linear axis: when length of travel is -400mm~+800mm, distance is 50mm:

Basic parameters set as follows:

1)Backward checking points: 8

2)Forward checking points: 16

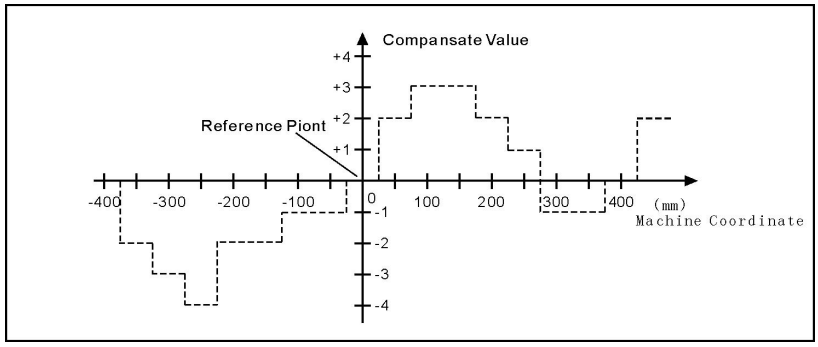
3)Multiple: 1

4)Distance(um): 50000

Corresponding compensation point and value:

No.	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	.	25
Value	+2	+1	+1	-2	0	-1	0	-1	+2	+1	0	-1	-1	-2	0	+1	+2		+1

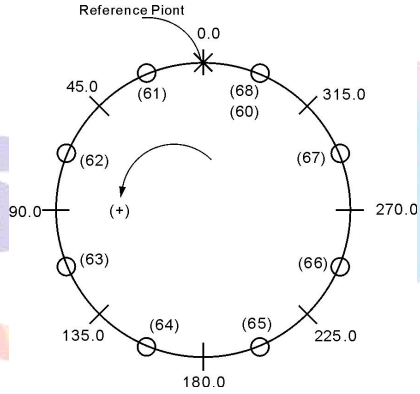
The contrasted chart of compensation points and value as follows:



Note: Zero point is reference point,don't account into checking point.

Example 2: Rotary axis: when movement per revolution is 360°,interval of points 45°,Basic parameters set as follows:

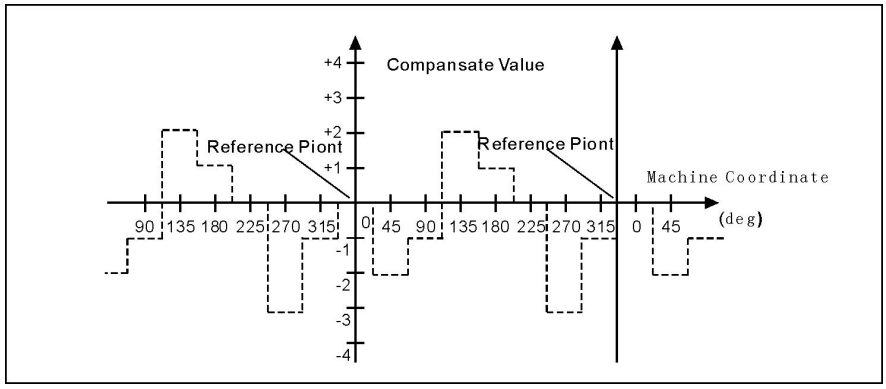
1) Backward checking points: 0
2) Forward checking points: 8
3) Multiple: 1
4) Distance(um): 45000



Output compensation value at corresponding point:

NO.	0	1	2	3	4	5	6	7	8
VALUE	+1	-2	+1	+3	-1	-1	-3	+2	+1

Compensation point and value contrast:



Note: 1. In the system, when axis is rotary axis, the coordinate value is 0~360. 0 and 360 are at same position: Example: When input A0 & A360 , both will rotate to zero position.

2.Zero point is reference point,don't account into checking point.

Chapter 3 Installation & Connection

3.1 System Installation

At first, users should check whether the hardware is complete, unwounded and compatible.

The installation of CNC system must be fastened tightly, with some spaces around to ensure the ventilation of air. Panel should be put in a place where it is not only convenient to operate and but also able to avoid hurt of heating by scrap iron.

Intense current, week current must be put separately, CNC system and driver should be possibly away from the machine intense current. In order to reduce interference, all signal cables should be kept away from AC contactor. Photoelectric encoder, limit, basic point signal are advisably not to be connected directly to CNC system through intense current box. All power cords must be earthing.

3.2 System installation dimension

This system has two parts of CNC controller, host controller & operational panel

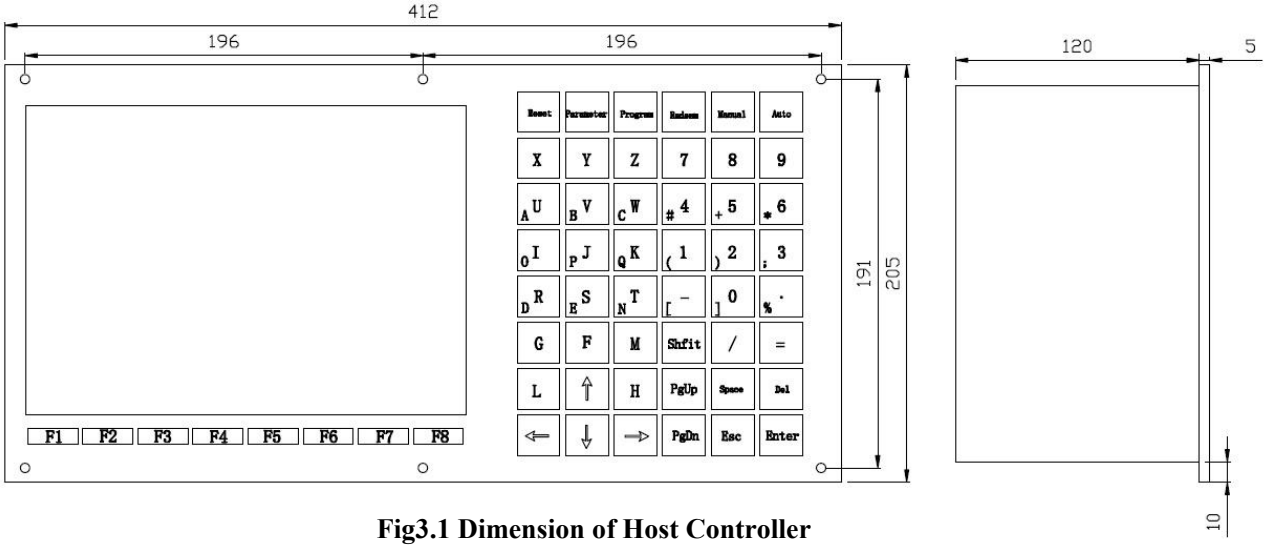


Fig3.1 Dimension of Host Controller

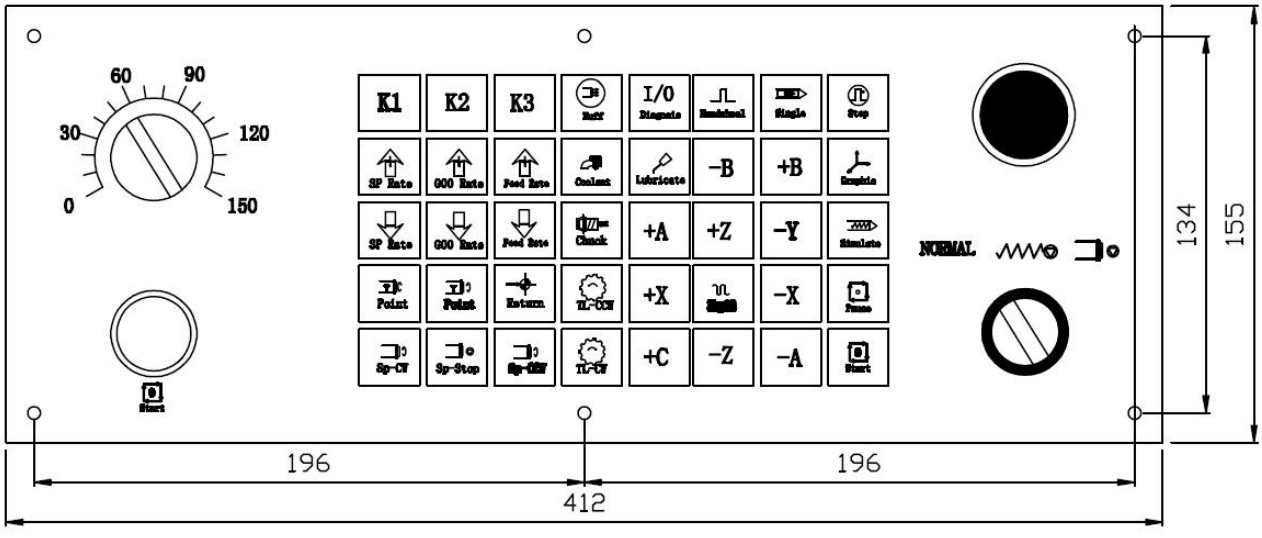


Fig3.2 Dimension of A type & B type Operational Panel

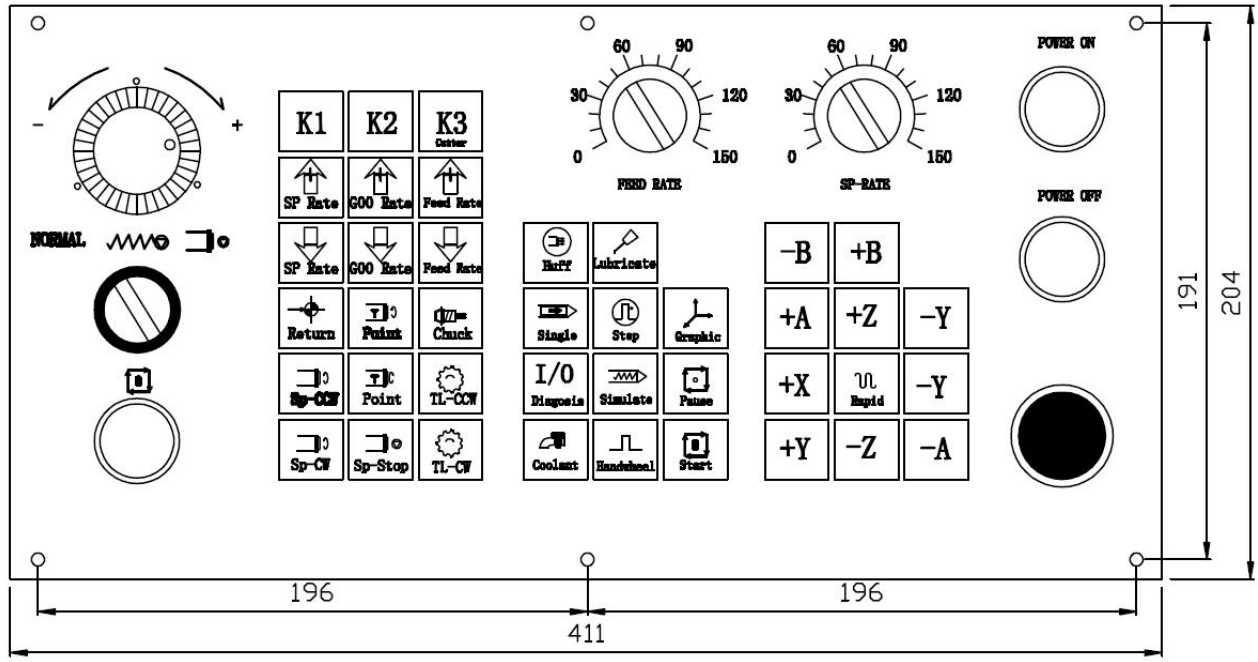
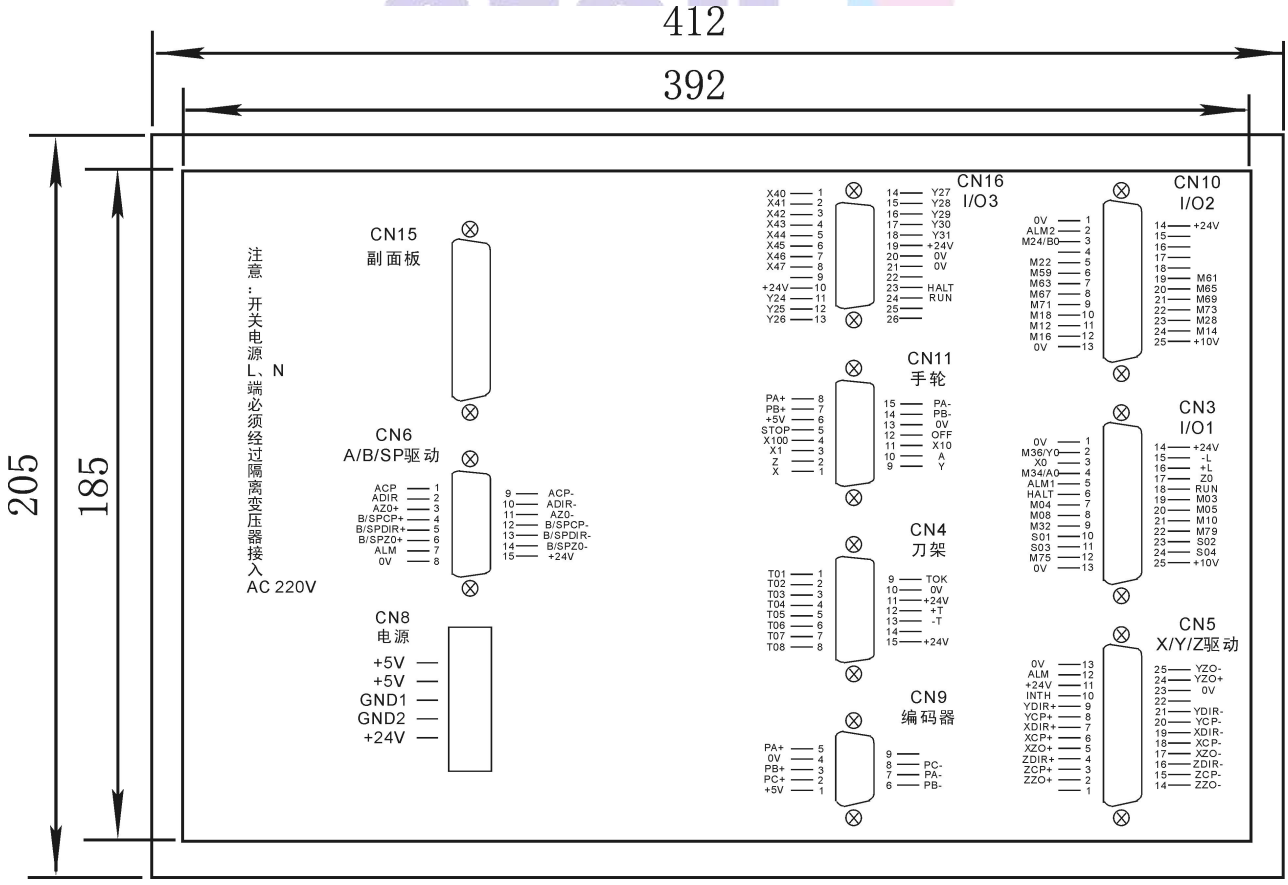


Fig3.3 Dimension of C type & E type Operational Panel

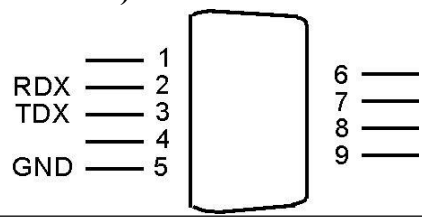
3.3 System Rear View



Attention: switching power supply L, N must be connected to AC 220V, current 0.5A through isolation transformer.

3.4 Interface Connection Graph

3.4.1 Communication Socket (Female/DB9)



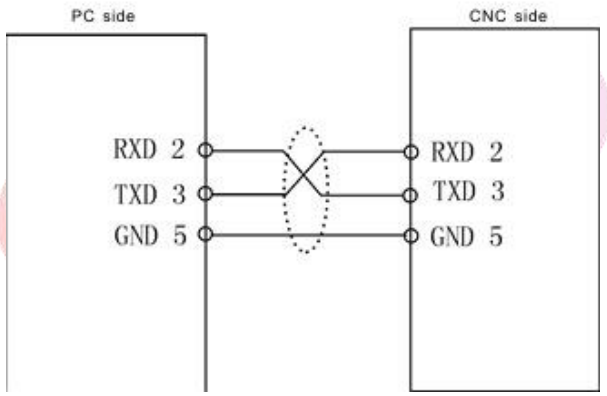
Communication signal with Female socket DB9				
signal	pin	I/O	function	Valid
0V	5	OUT	The ground of signal	0V
RXD	2	IN	The received data signal	
TXD	3	OUT	The transmission of data signal	

Note: 1. Connect to external PC with data communication, must be equipped with our special communication software, which is "SZGHCNCCS" software.

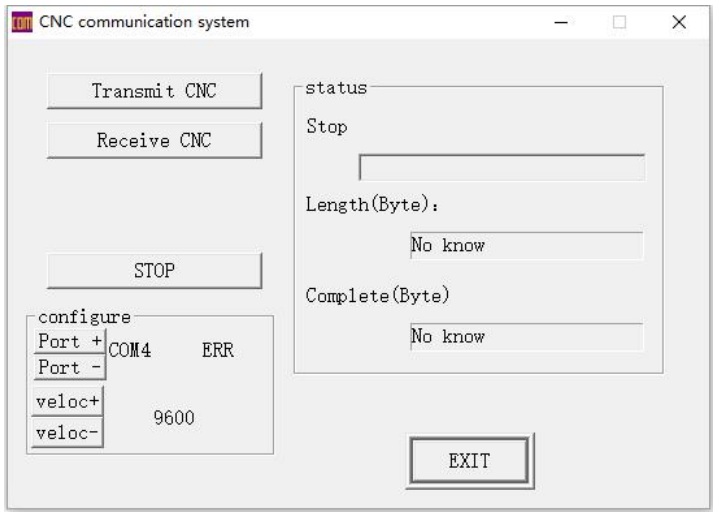
P37 in Other parameter is set rate of CNC system.

2.Communication line must adopt the shielded twisted pair cable, length shall not exceed 10m.

The signal of communication socket connect to PC:

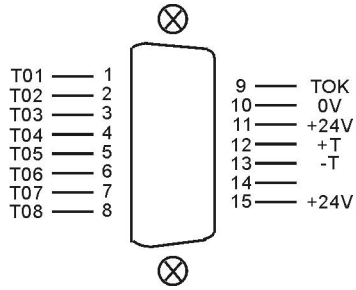


When PC programming, the files should be text files,which could be edit by Notepad or Word pad.



SZGHCNCCS software

3.4.2 CN4 Turret Socket (Female/DB15)

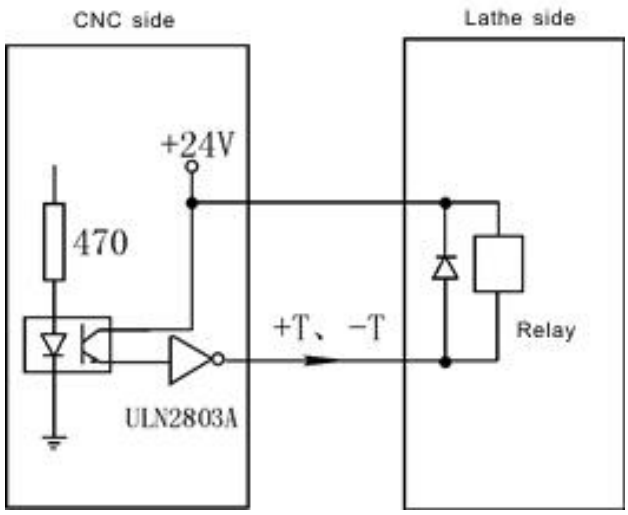


CN4 Turret signal with Female socket DB15				
Signal	Pin	I/O	function	Valid
0V	10	OUT	0V	0V
+24V	11,15	OUT	+24V	+24V
+T	12	OUT	Y18 Output	0V
-T	13	OUT	Y19 Output	0V
T1	1	IN	X0 Input Signal	0V
T2	2	IN	X1 Input Signal	0V
T3	3	IN	X2 Input Signal	0V
T4	4	IN	X3 Input Signal	0V
T5	5	IN	X4 Input Signal	0V
T6	6	IN	X5 Input Signal	0V
T7	7	IN	X6 Input Signal	0V
T8	8	IN	X7 Input Signal	0V
TOK	9	IN	X21 Input signal	0V

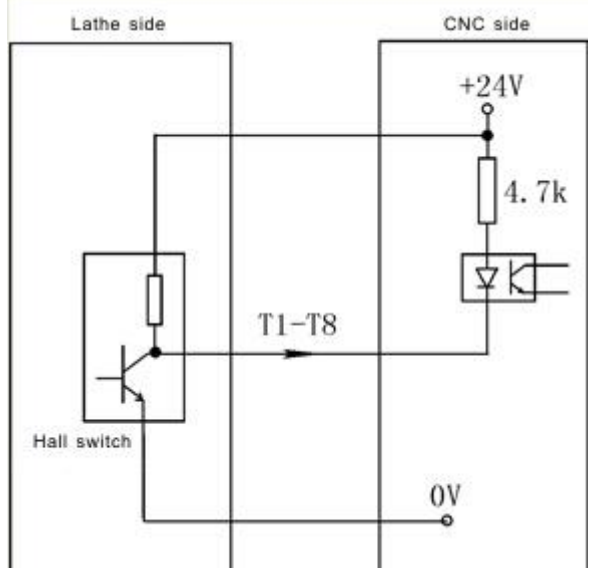
Attention: 1. All the input or output is for system, input is from exterior signal to system, the output is from system signal to external.

2. When choosing the electrical appliance plate, +T and –T control single contact middle relay, user should install two AC contactors of +T and –T.

System output signal +T/-T:

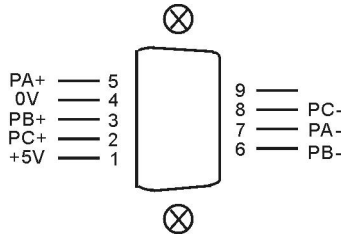


Tool input T1～T8、TOK signal



Attention:it must plus a reverse diode in order to cancel reverse current.

3.4.3 CN9 Spindle Encoder Socket (Female/DB9)

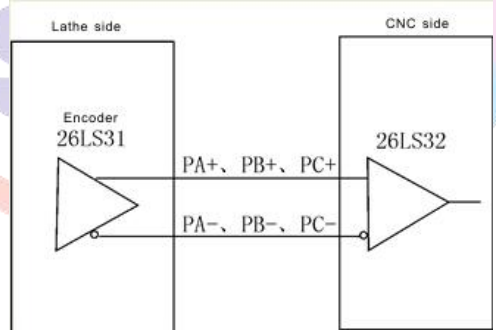


CN9 SP_Encoder signal with Female socket DB9				
Signal	Pin	I/O	Function	Valid
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	A Phase Positive signal	5V
PA-	7	IN	A Phase Negative signal	
PB+	3	IN	B Phase Positive signal	5V
PB-	6	IN	B Phase Negative signal	
PC+	2	IN	Z Phase Positive signal	5V
PC-	8	IN	Z Phase Negative signal	

Attention:

1. ***The output signal of encoder adopt the output way is line output, the power supply is +5V.***
2. ***The signal line must adopt shielded twisted pair cable, the length is 20m at most.***

The input signal of encoder PA PB PC:



Pay attention:

When machine is configured with inverter+ac motor and customer want to do some special processing,like G84, it needs to fix an encoder to spindle motor.

P412: number of spindle teeth

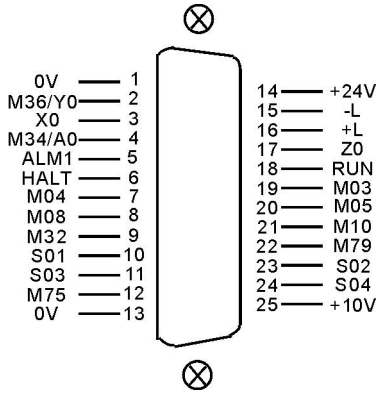
413, number of encoder teeth

When transmission ratio of spindle and encoder not as 1:1, please modify P412&P413 in Axis parameter when teeth of spindle is not more than teeth of encoder;

If teeth of spindle is more than teeth of encoder, it needs to select adapter plate of SZGH;

Note: it must be integer multiple relationship about teeth between spindle & encoder.

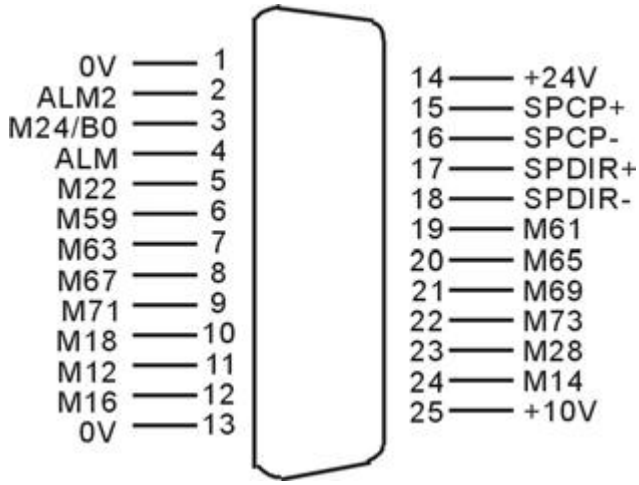
3.4.4 CN3 IO1 Control Socket (Female/DB25)



CN3 I/O1 signal with Female Socket of DB25				
signal	pin	I/O	function	Valid
0V	1	OUT	0V	0V
+24V	14	OUT	+24V	+24V
M36/Y0	2	IN	Zero Point of Y-axis	0V
X0	3	IN	Zero Point of X-axis	0V
Z0	17	IN	Zero Point of Z-axis	0V
-L	15	IN	Positive limit	0V
+L	16	IN	Negative limit	0V
M34/A0	4	IN	M34/Zero Point of A-axis	0V
ALM1	5	IN	Alarm1 of Spindle	0V
HALT	6	IN	Pause	0V
RUN	18	IN	Run	0V
M03	19	OUT	Clockwise Rotation of Spindle	0V
M04	7	OUT	Counter clockwise Rotation of Spindle	0V
M05	20	OUT	Stop of Spindle	0V
M08	8	OUT	Coolant	0V
M10	21	OUT	Chuck	0V
M32	9	OUT	Lubrication	0V
M79	22	OUT	User-defined output8	0V
S01	10	OUT	Spindle first gear	0V
S02	23	OUT	Spindle second gear	0V
S03	11	OUT	Spindle third gear	0V
S04	24	OUT	Spindle fourth gear	0V
M75	12	OUT	User-defined output9	0V
+10V	25	OUT	Analog Output Signal of 1st spindle	0～10V
0V	13	OUT	Ground of frequency conversion	0V

Note: when your CNC system isn't configured with A-axis,M34/A0 could be used as input point,controlled by M34 code.

3.4.5 CN10 IO2 Socket (Female/DB25)



CN10 I/O2 signal with Female Socket of DB25				
Signal	Pin	I/O	Function	Valid
0V	1	OUT	Ground of the power supply	0V
+24V	14	OUT	24V power supply	+24V
ALM2	2	IN	Alarm2 of Machine Tool	0V
M24	3	IN	User-defined input 7	0V
M22	5	IN	M01 Quasi-stop Input	0V
M59	6	OUT	Huff Output	0V
M61	19	OUT	User-defined output1	0V
M63	7	OUT	User-defined output2	0V
M65	20	OUT	User-defined output3	0V
M67	8	OUT	User-defined output4	0V
M69	21	OUT	User-defined output5	0V
M71	9	OUT	User-defined output6	0V
M73	22	OUT	User-defined output7	0V
M18	10	IN	User-defined input1	0V
M28	23	IN	User-defined input2	0V
M12	11	IN	User-defined input3	0V
M14	24	IN	User-defined input4	0V
M16	12	IN	User-defined input5	0V
+10V	25	OUT	Analog Voltage of 2nd Spindle	0～10V
0V	13	OUT	Ground of frequency conversion	0V

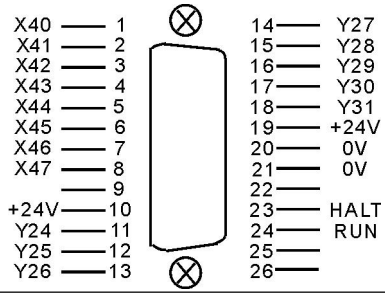
Special application for spindle system

Signal	Pin	I/O	Function	Valid
SPCP+	16	OUT	Positive Pulse signal of SP-axis	5V
SPCP-	17	OUT	Negative Pulse signal of SP-axis	
SPDIR+	18	OUT	Positive Direction signal of SP-axis	5V
SPDIR-	19	OUT	Negative Direction signal of SP-axis	

Note: 1.This axis belongs extra axis(+1), cannot interpolation with another axes at same time. It only could be used for control position/degrees of spindle servo.

2. User need this functions, please remark this with us.

3.4.6 CN16 IO3 Socket (Female/DB26)



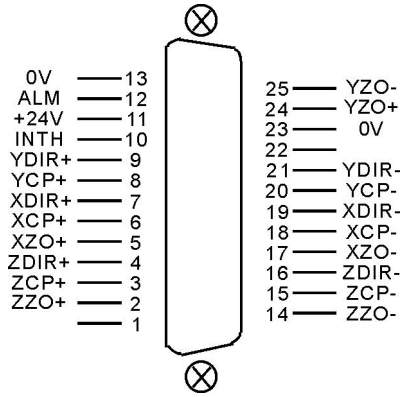
CN16 I/O3 signal with Female Socket of DB25				
Signal	Pin	I/O	Function	Valid
0V	20 , 21	OUT	0V	0V
+24V	10 , 19	OUT	+24V	+24V
X40	1	IN	Alternate input 0	0V
X41	2	IN	Alternate input 1	0V
X42	3	IN	Alternate input 2	0V
X43	4	IN	Alternate input 3	0V
X44	5	IN	Alternate input 4	0V
X45	6	IN	Alternate input 5	0V
X46	7	IN	Alternate input 6	0V
X47	8	IN	Alternate input 7	0V
Y24	11	OUT	Alternate output 0	0V
Y25	12	OUT	Alternate output 1	0V
Y26	13	OUT	Alternate output 2	0V
Y27	14	OUT	Alternate output 3	0V
Y28	15	OUT	Alternate output 4	0V
Y29	16	OUT	Alternate output 5	0V
Y30	17	OUT	Alternate output 6	0V
Y31	18	OUT	Alternate output 7	0V
RUN	24	IN	External Input for Run	0V
HALT	23	IN	External Input for Halt	0V

Note: 1.Y24 is controlled by K1 key on operational panel.

2.Y25 is controlled by K2 key on operational panel.

3.Y26 is controlled by K3 key on operational panel.

3.4.7 CN5 XYZ Drive Socket (Male/DB25)



CN5 XYZ Driver with Male Socket of DB25				
Signal	Pin	I/O	Function	Valid
XCP+	6	OUT	Positive Pulse signal of X-axis	5V
XCP-	18	OUT	Negative Pulse signal of X-axis	
XDIR+	7	OUT	Positive Direction signal of X-axis	5V
XDIR-	19	OUT	Negative Direction signal of X-axis	
YCP+	8	OUT	Positive Pulse signal of Y-axis	5V
YCP-	20	OUT	Negative Pulse signal of Y-axis	
YDIR+	9	OUT	Positive Pulse signal of Y-axis	5V
YDIR-	21	OUT	Negative Pulse signal of Y-axis	
XZO+	5	IN	Positive Zero position signal of X-axis	5V
XZO-	17	IN	Negative Zero position signal of X-axis	
ZCP+	3	OUT	Positive Pulse signal of Z-axis	5V
ZCP-	15	OUT	Negative Pulse signal of Z-axis	
ZDIR+	4	OUT	Positive Direction signal of Z-axis	5V
ZDIR-	16	OUT	Negative Direction signal of Z-axis	
ZZO+	2	IN	Positive Zero Position signal of Z-axis	5V
ZZO-	14	IN	Negative Zero Position signal of Z-axis	
YZ0+	24	IN	Positive Zero Position signal of Y-axis	5V
YZ0-	25	IN	Negative Zero Position signal of Y-axis	
0V	13 , 23	OUT	0V	5V
ALM	12	IN	Alarm signal of Servo driver	
+24V	11	OUT	+24V of Power Supply	0V
INTH	10	OUT	Reset alarm signal	0V

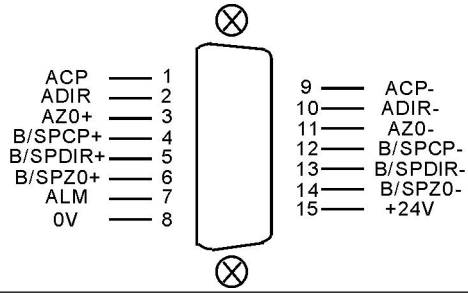
Note: 1. The signal cables must adopt shielded twisted pair cable, the length is 20m at most.

2. Whether the alarm signal ALM is normal open or normal close is set by P17 in Other parameter.

3. Control signals for Y-axis also is same to control signals for C axis.

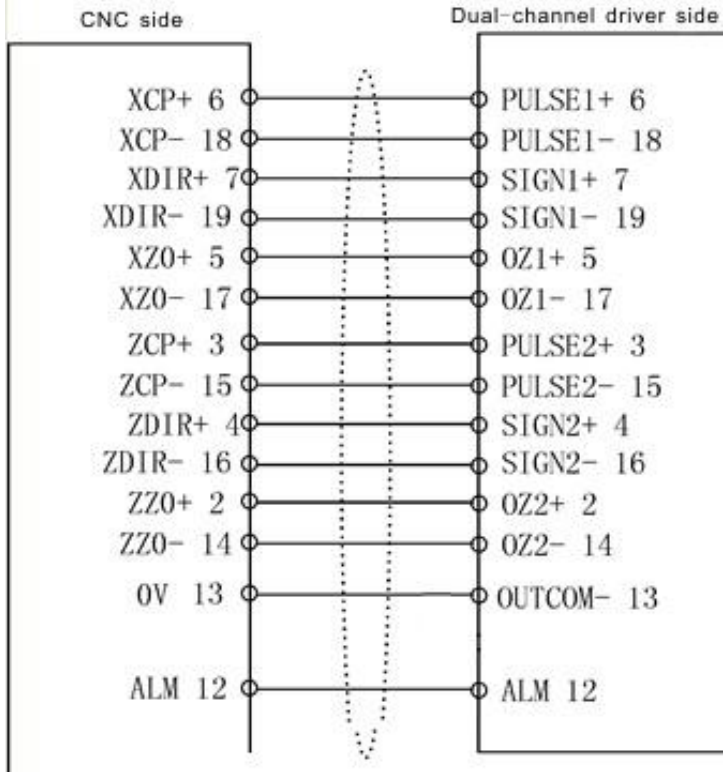
4. When system take C axis to as rotate axis, M800 instruction is for backing to zero position of encoder, Output M75 signal to select position control mode of spindle servo, M03/M04 is to close M75 signal, spindle servo shift to speed control mode.

3.4.8 CN6 AB Drive Socket (Male/DB15)

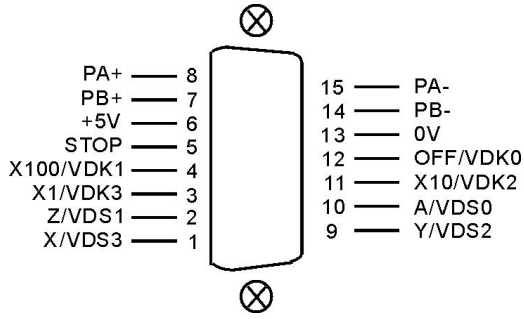


CN6 AB Driver with Male Socket of DB15				
Signal	Pin	I/O	Function	Valid
ACP+	1	OUT	Positive Pulse signal of A-axis	5V
ACP-	9	OUT	Negative Pulse signal of A-axis	
ADIR+	2	OUT	Positive Direction signal of A-axis	5V
ADIR-	10	OUT	Negative Direction signal of A-axis	
BCP+	4	OUT	Positive Pulse signal of B-axis	5V
BCP-	12	OUT	Negative Pulse signal of B-axis	
BDIR+	5	OUT	Positive Pulse signal of B-axis	5V
BDIR-	13	OUT	Negative Pulse signal of B-axis	
AZO+	3	IN	Positive Zero position signal of A-axis	5V
AZO-	11	IN	Negative Zero position signal of A-axis	
BZO+	6	IN	Positive Zero position signal of B-axis	5V
BZO-	14	IN	Negative Zero position signal of B-axis	
0V	8	OUT	0V	0V
ALM	7	IN	Alarm signal of Servo driver	0V
+24V	15	OUT	+24V of Power Supply	24V

Example: Wiring Diagram[CNC Controller with dual servo drive(SZGH-302)]



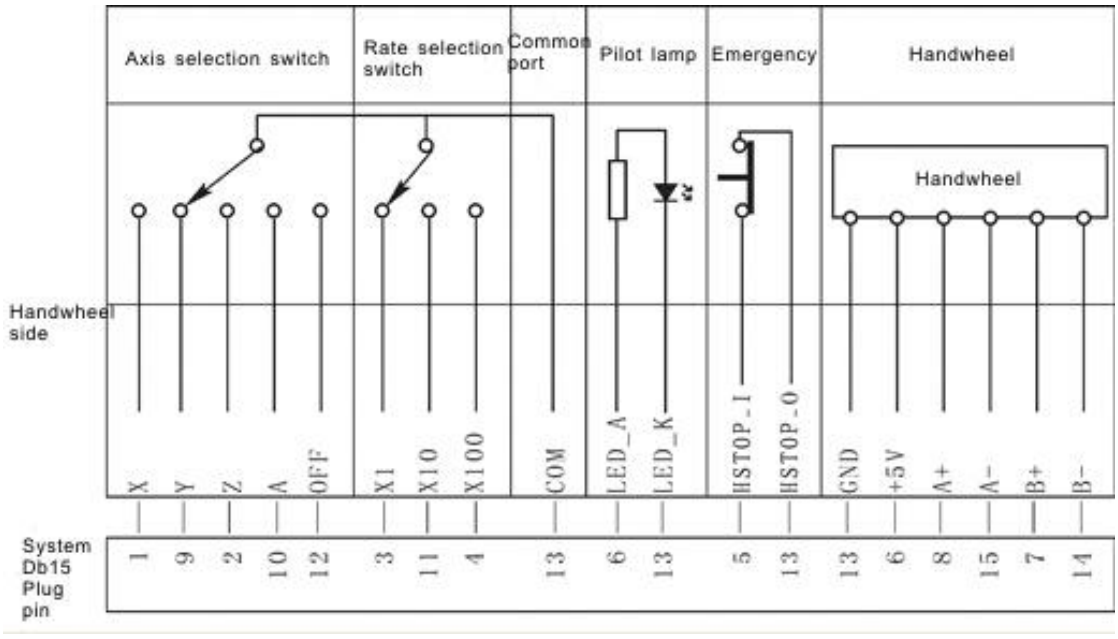
3.4.9 CN11 MPG/Handhold Box Socket (Male/DB15)



CN11 Handwheel Signal with Male Socket of DB15				
signal	pin	I/O	function	Availability
0V	13	OUT	0V	0V
+5V	6	OUT	+5V	+5V
PA+	8	IN	A signal +	5V
PA-	15	IN	A signal -	
PB+	7	IN	B signal +	5V
PB-	14	IN	B signal -	
STOP	5	IN	emergency stop	0V
OFF/VDK0	12	IN	Off/ feed amending 0	0V
X100/VDK1	4	IN	*100/ feed amending 1	0V
X10/VDK2	11	IN	*10/ feed amending 2	0V
X1/VDK3	3	IN	*1/ feed amending 3	0V
A/VDS0/HALT	10	IN	A/SP amending 0/halt stop	0V
Z/VDS1	2	IN	Z/SP amending 1	0V
Y/VDS2/RUN	9	IN	Y/SP amending 2/run	0V
X/VDS3	1	IN	X/SP amending 3	0V

3.4.9.1 Electrical handwheel (Manual pulse generator)

Handwheel contact diagrammatic as:

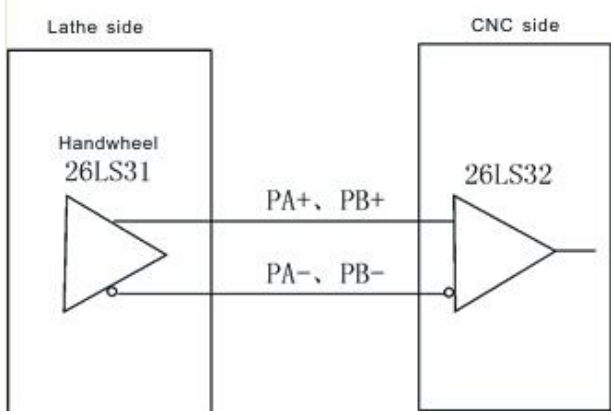


When user connect our handhold box to CN11 Plug, P1 in Other parameter needs to set 1,

and cannot use band switch to adjust SP_Rate, Feed_Rate & External Run/Pause, and P1&P2 in Axis parameter only could be “0”. X Y Z A X1 X10 X100 inputs are for axis-selection & rate, P33&P34 in Other parameter only could set to 0.

PA+ PB- PA+ PA- are corresponding input signal of handwheel pulse A B.

The input signal of handwheel:



Attention:

1. The output signal of handwheel adopts line output, the power supply is +5V.

2. Just connect PA+ PB+ if adopt voltage output.

3. Manual pulse generator needn’t switch button for Enter ON/OFF handwheel, if there is a switch for Enter, it is okay that use short connection of switch.

3.4.9.2 Using for Band Switch

When P1 & P2 in Axis parameter is set to “1”, VDK0/VDK1/VDK2/VDK3 & VDS0/VDS1/VDS2/VDS3 are working, which can’t as inputs for external Run/Halt button, P1 in Other parameter is 0;

VDS0(A) VDS1(Z) VDS2(Y) VDS3(X) are the input signal of adjust rate of spindle, total 16 gears. VDK0(OFF) VDK1(X100) VDK2(X10) VDK3(X1) are inputs signal of adjust Rate of Feeding speed, total 16 gears.

3.4.9.3 External Switch for Run/Halt

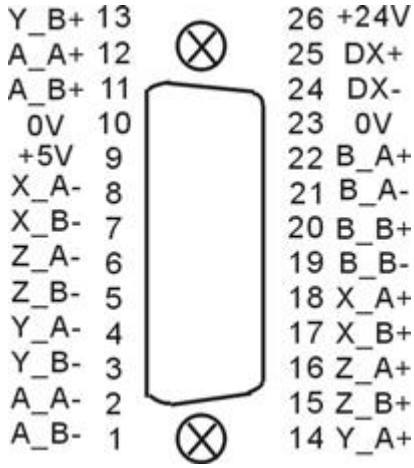
When P33 in Other parameter is “1”, PIN9 of CN11 plug can be as input for external Run , which running program automatically; When P34 in Other parameter is set to “1”, PIN10 of CN11 plug can be as input for external Halt, which pause processing program.

3.4.9.4 Using for External Emergency Stop

STOP signal is the input signal of external emergency button, P27 in Other parameter is set for type of switch of emergency stop button. 0: NO type, 1: NC type.

Suggestion: Configured with SZGH Handhold box(MPG), which is better to operate SZGH CNC system, please check 1.5.3_Book2.

3.4.10 CN13 Position Feedback Socket(Male/DB26)



CN13 Position Feedback with DB26 Male Socket				
Signal	Pin	I/O	Function	Valid
0V	10,23	OUT	Ground of power supply	0V
+5V	9	OUT	5V power supply	+5V
+24V	26	OUT	24V power supply	+24V
DX+	25		RS485+	
DX-	24		RS485-	
XA+	18	IN	Positive signal A of X axis	5V
XA-	8	IN	Negative signal A of X axis	
XB+	17	IN	Positive signal B of X axis	5V
XB-	7	IN	Negative signal B of X axis	
YA+	14	IN	Positive signal A of Y axis	5V
YA-	4	IN	Negative signal A of Y axis	
YB+	13	IN	Positive signal B of Y axis	5V
YB-	3	IN	Negative signal B of Y axis	
ZA+	16	IN	Positive signal A of Z axis	5V
ZA-	6	IN	Negative signal A of Z axis	
ZB+	15	IN	Positive signal B of Z axis	5V
ZB-	5	IN	Negative signal B of Z axis	
AA+	12	IN	Positive signal A of A axis	5V
AA-	2	IN	Negative signal A of A axis	
AB+	11	IN	Positive signal B of A axis	5V
AB-	1	IN	Negative signal B of A axis	
BA+	22	IN	Positive signal A of B axis	5V
BA-	21	IN	Negative signal A of B axis	
BB+	20	IN	Positive signal B of B axis	5V
BB-	19	IN	Negative signal B of B axis	

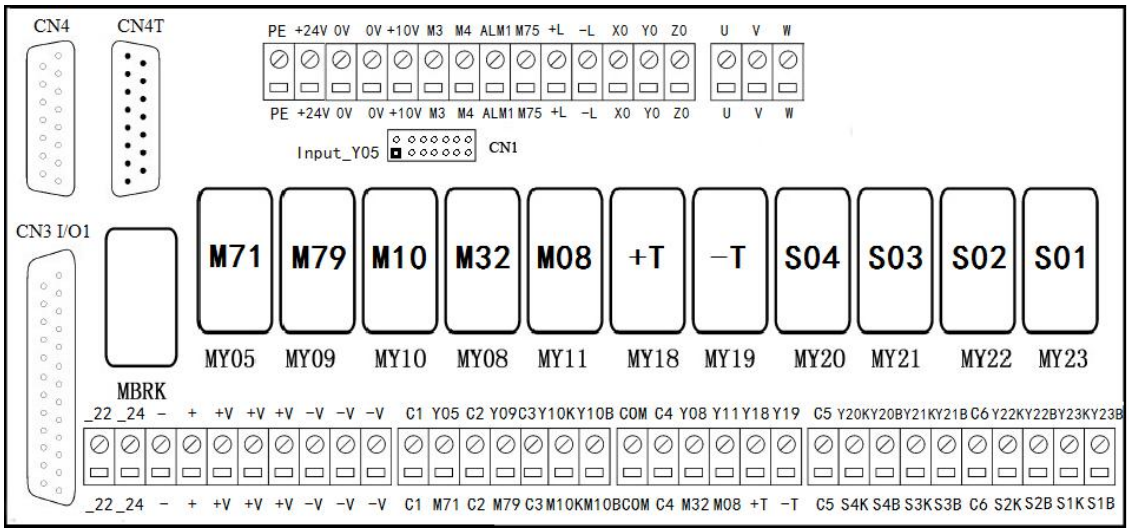
P200-P220 in Other parameter set the function. Press "G" in Diagnosis to clear the instruction position and feedback position and clear the deviation alarm after alarm.

Attention: 1, The encoder or the grating output signal with long-line output mode (also RS422), the power supply is +5V.

2, The signal line must adopt shielded twisted pair cable, the length shall not exceed 20m.

3.5 I/O Relay Board

I/O Relay board is optional item,its model is NEWPORT-12,with 12pcs of relays, which is used for connecting CNC controller and external switches & loads easily.



CN3_I/O1 socket is corresponding to CN3 plug of CNC Controller one by one;

CN4 socket is corresponding to CN4 plug of CNC Controller one by one;

CN1 pin output(spare input for control relays): Input_Y05 is Pin1_CN1 plug

Pin1_CN1 to Pin10 is spare input points for control relays,MY05,MY08,MY09,MY10,MY11, MY18,MY19,MY20,MY21,MY22,MY23 .

Note: 1.Pin6(Halt) & Pin18(Run) on CN3 plug are not connected.

2.Pin9(Tok) on CN4 plug is not connected.

3.5.1 Control of Turret (CN4)

CN4T is connected to position signal of turret on machine tool and also supply power to encoder/sensors on turret.

Pin	1	2	3	4	5	6	7	8	9	10/15
Func.	T1	T2	T3	T4	T5	T6	T7	T8	0V	+24V
Add.	X0	X1	X2	X3	X4	X5	X6	X7	-	-

C4 is common port 4 of +T, -T and M08 & M32.

M08 is for control cooling device.

M32 is for control lubricate device.

U	Connecting to power of turret,which is used for filter
V	
W	
COM	Common port for filter circuit

Suggestion: use +T/-T to control AC contactor for CW/CCW of Turret.

3.5.2 Control of Spindle (CN3)

Remark	Pin	Function
+24V	P14_CN3	Positive Port of Power
0V	P13_CN3	Negative Port of Power
+10V	P25_CN3	Analog Voltage Output
M3	P19_CN3	CW Control for Spindle
M4	P7_CN3	CCW Control for Spindle
ALM1	P5_CN3	Spindle Alarm Input
M75	P12_CN3	Spindle Position Control Output
C5	-	Common Port 5 for S3 & S4 Gear Output
S4K	-	Normal Close type Output
S4B	-	Normal Open type Output
S3K	-	Normal Open type Output
S3B	-	Normal Close type Output
C6	-	Common Port 6 for S1&S2 Gear Output
S2K	-	Normal Close type Output
S2B	-	Normal Open type Output
S1K	-	Normal Open type Output
S1B	-	Normal Close type Output

Note: 1.NC output or NO output is based on C5/C6 common port.

2. M3,M4,M75 are output from CNC controller directly without through relay, if control high voltage devices, it needs to add relays/contactors with them.

3.5.3 Control of Braker (Servo)

Mark	Pin	Function
_24	P24_Driver	Positive of Braker Control Port on SZGH Servo Driver
_22	P22_Driver	Negative of Braker Control Port on SZGH Servo Driver
-	P2_Motor	Negative of Braker on SZGH Servo Motor
+	P1_Motor	Positive of Braker on SZGH Servo Motor
+V	-	Positive Port of Braker Power
-V	-	Negative Port of Braker Power

PIN24 & PIN22 on CN3 control plug of SZGH series Servo Driver are connected to _24 & _22 of IO control board directly; - & + are connected to Braker of servo motor directly; +V & -V are port for input power of braker power.

3.5.4 Define of I/O (Other)

Remark	Pin	Function
-L	P15_CN3	Limit in negative direction of all axes
+L	P16_CN3	Limit in positive direction of all axes
X0	P3_CN3	X_Homing Switch Input
Y0	P12_CN3	Y_Homing Switch Input
Z0	P17_CN3	Z_Homing Switch Input
M79	P22_CN3	Tailstock/User-Defined Output
Input_Y05	P9_CN10	M71 command Output
M08	P8_CN3	Coolant Control Output
M32	P9_CN3	Lubricate Control Output

Note: 1. If user needs use MY05 relay,please connecting M71 on CN10 to Input_Y05 on IO relay Board, if for control chuck, please reference Chuck part on User manual.

2.When output points is with relay, user can use them to control loads directly; when output points is output from CNC controller directly without relay, it cannot control external devices directly, otherwise it will damage CNC controller.

3. When power of controlling devices is over 250VAC/12A, please add contactors.

4. Valid level of all inputs & outputs of SZGH CNC controller is 0V.

5. When without Y-axis/C-axis, Y0 is input point of M36 command.

6. M79 is for tailstock on Lathe system, user-defined output on Milling system.

3.6 Daily Maintenance and Repair

In order to plenty use CNC system's function and promote efficiency,the most important work is correctly using system， and notice system's daily maintenance work， promote Mean Time Between Failures MTBF.Now this system's maintenance method is introduced as follows：

3.6.1 Maintain

System's using must be under the good circumstance.

Operator,programmer and repairer must be familiar with NC machining technology, and according the require of user book correctly use，do one's best to avoid improper operation.

Everyday operator should clean the system's box and panel in case for corrupt thing and sundries to indemnify it.

When CNC system's using time is over three month,operator should open the system box and clean inside.

If not using system for long time,should boot the system one time every week.

3.6.2 Ordinary Problem

3.6.2.1 System can't boot

1） check if input power is normal.
2） check if power switch is turn on.
3） check insurance.

3.6.2.2 No display as boot

1） Boot again or reset.
2） Check if switch power's ＋5V、＋12V、－12V、-24V are normal.
3） Check if transformer is bad.
4） Check if LCD's bright adjust and connection are normal.
5） Check if main board is normal.

3.6.2.3 System's control disorganize

1） Wrong operations.
2） Anti-jamming ability of power supply is descend.
3） Working circumstance of CNC system is too bad.

3.6.2.4 Lose of user program

The DC battery on system main board can insure user's program and parameter don't lose.When system isn't used for half year or system has been used for over two years，the battery maybe invalidate,therefore, should exchange battery.

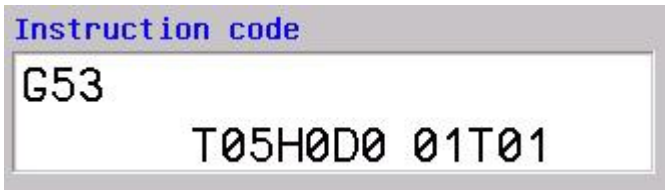
3.6.2.5 Machining precision is bad

1） CNC Machine needs to revise backlash after using some time.
2） Best to revise base point before machining in order to insure the start point's precision.
3） Machining speed and cutting depth is improper.
4） Machine connector's prick melt falls off.
5） Tool isn't tightened.
6） Piece clamp isn't good.
7） Tool's giving up isn't equality because piece's dimension isn't uniformity.
8） Problems of machine Tool

Attention: Because of many kinds of reasons this Manual book may have some mistakes. We will appreciate that you feedback to us or our agents , to provide services and technical support for every customer.

Chapter 4 CNC Machining Center

4.1 Status of CNC Machine Center System



“ Tab HxDx cdTef ” status in the system,
“ ab ” : Number of tool in spindle, Eg: 05 means No.5 tool in spindle
“ cd ” : Number of current tool holder in tool magazine
“ ef ” : Number of current tool in tool magazine
H0: current tool length compensation ; D0: current tool radius compensation

4.2 Umbrella Tool Magazine

4.2.1 I/O ports for Umbrella Tool Magazine

Signal	PIN	I/O	Function
T01	P1_CN4	Input	Detecting if overload of magazine’s motor
T02	P2_CN4	Input	Detecting if position of Chuck loosing tool
T03	P3_CN4	Input	Detecting if level of lubricating oil is low
T04	P4_CN4	Input	Detecting if coolant water is lacking
T05	P5_CN4	Input	Detecting if position of Chuck tightening tool
T06	P6_CN4	Input	Detecting if position of tool magazine forward
T07	P7_CN4	Input	Detecting if position of tool magazine backward
T08	P8_CN4	Input	Counting of tools
ALM1	P5_CN3	Input	Alarm input of Spindle Driver
ALM2	P2_CN10	Input	Detecting if overload of cooling motor
M22	P5_CN10	Input	Detecting if spindle orientation end
M12	P11_CN10	Input	Detecting if safe door of machine tool is open
M14	P24_CN10	Input	Detection for pressure alarm of compressed air
M16	P12_CN10	Input	External Switch for SP loosen tool
X40	P1_CN16	Input	Detecting if overload of motor for removing chips
X46	P7_CN16	Input	Activate M01 code(exact stop)
X47	P8_CN16	Input	Detecting if overload of fan of spindle
M61	P19_CN10	Output	Spindle orientation
M63	P7_CN10	Output	Rotate CW of tool magazine
M65	P20_CN10	Output	Yellow indicator lamp (Stop)
M67	P8_CN10	Output	Red indicator lamp (Fault)
M69	P21_CN10	Output	Green indicator lamp (Running)
M71	P9_CN10	Output	Rotate CCW of tool magazine
M73	P22_CN10	Output	Forward of tool magazine

M59	P6_CN10	Output	Huff/Blower
M32	P9_CN3	Output	Lubrication
M10	P21_CN3	Output	Spindle chuck loose tool
M08	P8_CN3	Output	Coolant
M05	P20_CN3	Output	Stop spindle/Reset M03&M04
M04	P7_CN3	Output	Rotate CCW of Spindle
M03	P19_CN3	Output	Rotate CW of Spindle
M75	P12_CN3	Output	Shift control mode(Position/Speed) of spindle system
Y24	P11_CN16	Output	Backward of tool magazine
Y28	P15_CN16	Output	M203: CW_2nd Spindle
Y29	P16_CN16	Output	M204: CCW_2nd Spindle
Y30	P17_CN16	Output	SPEN: Output Enable of SP servo driver
Y31	P18_CN16	Output	Output Enable of 2nd SP servo driver

Note: 1. Tool magazine must backward at position when turn on spindle, also T07 is valid.

2. When M12 is as input point for detecting safe door,it needs to set P7=1 on Other parameter.

3. When M16 is as input point for control chuck loose tool, it needs to set P22=1 on Other parameter.

4.2.2 Parameters for Umbrella Tool Magazine

(a) Tool parameter for umbrella tool magazine

P10=1, Activate the program of tool change;

P11=0, Only Z-axis move to position of tool change;

P12=2, Feeding axes move to position of No.2 reference point when changing tool;

P13=1, Detect SP-orientation(M22) and Feeding axis move to reference point position;

P14=1, Spindle do orientation when changing tool (M61);

P15=2, Type of tool magazine is umbrella type;

P16=0, Standard type tool magazine;process of tool change is as chapter4.2.4.

P17=120, Lifting height of Z axis when changing tool, increment value (unit:mm);

P18=2000, Lifting speed of Z axis when changing tool (unit:mm/min);

P19=5, Delay time before magazine backward to home (unit: 0.1 sec), also after T06&T05 are valid, also detecting that position of magazine forward & spindle tighten tool.

P20=5, Delay time after loosening tool when changing tool(unit: 0.1 sec) ,also after T02 is valid.

P22=1, Interlock between motion of Z-axis and umbrella tool magazine[1:Yes, 0:No], set to "1" means Z-axis can move after detecting T07 is valid,also position of magazine backward;

P23=1, Detect position of Z-axis when use magazine forward command:M71 [0:Yes, 1:No], when “P23=0”, System allow exchange tool when current coordinate of Z axis is higher 20mm than position of tool change (it is used for debugging);

P25=1007 , X07 is input point for counting tools,Valid level is rising along: “1000+ number”, [if valid level is falling along: “2000+ number”];

P27=1001, Y01 is the output point for control rotation CW of tool magazine [1000+number];

P28=1005, Y05 is the output point for control rotation CCW of tool magazine [1000+number];

P100=0, reference point 1 X (mm) ;

P101=0, reference point 1 Y (mm) ;

P102=0, reference point 1 Z (mm) ;

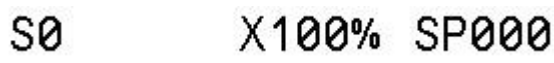
P103=0, reference point 1 A (mm) ;

P104=0, reference point 2 X (mm) ;
P105=0, reference point 2 Y (mm) ;
P106=50, reference point 2 Z (mm) ;
P107=0, reference point 2 A (mm) ;

(b) Axis Parameter for offset of spindle orientation

1) P400=1, detect position feedback of spindle when changing tool, set to "0": no detect ;

2) P401, Detecting degree when spindle orientation, manually press K1,after finishing spindle orientation, input value of SP into P401; Eg: current value of SP is 000 as following picture show:



3) P402, Allow error of detecting degree when spindle orientation;

Note: encoder signal of spindle must be connected to system(CN9 plug).

4.2.3 Debug of Umbrella Tool Magazine

1) Operation of Manual Buttons

K1: Spindle orientation (Indicator of K1 is on after orientation end): Output M61, detect M22 ;
K2: Z-axis move to point of tool changing ;
K3: Setting number of current tool holder;

"**TL-CW**":Magazine rotate CW, Output M63,Pin7_CN10 Plug,detect input of T08 ;

"**TL-CCW**":Magazine rotate CCW,Output M71,Pin9_CN10 Plug,detect input of T08 ;

2) M Instruction

M71: Forward of magazine,reset Y24, output M73,Pin22_CN10 Plug;
M73: Backward of magazine,reset M73, output Y24, Pin11_CN10 Plug;
M881: Spindle Orientation , same function as "K1 key";

3) Instruction of Tool Change

Txx: Return tool of spindle back to magazine & change Txx to spindle; Eg: ***T02 , also No.2 tool.***

M36 Txx: Change Txx to spindle in single step mode. After finish one step,system will be suspended, Press “Start”button again to execute the next step(for debugging).

4.2.4 Process of Tool Changing

① Feeding axes move to position of tool changing,also position of reference point ;
② Spindle orientation: output M61(P7_CN10 plug), detect M22 (orientation end);
③ Magazine goes forward:output M73(P22_CN10 plug),detect T06 (position of forward);
④ Chuck loose tool: output M10(P10_CN3 plug), detect T02(Position of loose tool);
⑤ Z axis lifting, height is set by P17 on Tool parameter;
⑥ Magazine rotating CW/CCW: output M63/M71, detect T08(count of tools);
⑦ Chuck loose tool: output M10, detect T02(Position of SP-Loose tool);

⑧ Z axis falls down to point of tool changing ;
⑨ Chuck tighten tool: reset M10,detect T05(Position of tightening tool);
⑩ Tool magazine backward: output Y24, detect T07(Position of backward);

Special attention:

1. Feeding axes must go home before tool change.

2. No.7 spindle-chuck loose tool & detect position of loose tool, which ensure without tool on chuck.

3. When power off suddenly or emergency stop was happening during changing tool,which cause the tool case is in a mess , please rotate several tool position of magazine manually and then use K3 to set the number of current tool holder;

4. Pay attention to check position of Z axis in case of accident when using instruction M71 to make tool magazine going forward;

5. The position of No 0 tool (T0) should not have tool, otherwise perhaps occur accident;

6. When install tool at first time, there must be a tool in the spindle (if CNC display T00, after initialize the table of tool case usually, No.1 tool should be at spindle).

4.3 Standard Arm Type Tool Magazine

4.3.1 I/O ports for Arm type Tool Magazine

Signal	PIN	I/O	Function
T01	P1_CN4	Input	Detecting if overload of magazine's motor
T02	P2_CN4	Input	Detecting if position of Chuck loosing tool
T03	P3_CN4	Input	Detecting if level of lubricating oil is low
T04	P4_CN4	Input	Detecting if coolant water is lacking
T05	P5_CN4	Input	Detecting if position of Chuck tightening tool
T08	P8_CN4	Input	Counting of tools
ALM1	P5_CN3	Input	Alarm input of Spindle Driver
ALM2	P2_CN10	Input	Detecting if overload of cooling motor
M22	P5_CN10	Input	Detecting if spindle orientation end
M12	P11_CN10	Input	Detecting if safe door of machine tool is open
M14	P24_CN10	Input	Detection for pressure alarm of compressed air
M16	P12_CN10	Input	External Switch for SP loosen tool
X40	P1_CN16	Input	Detecting if overload of motor for removing chips
X41	P2_CN16	Input	Detecting if position of tool holder
X42	P3_CN16	Input	Detecting if position of arm(take/exchange/return)
X44	P5_CN16	Input	Detecting if position of tool holder lift up
X45	P6_CN16	Input	Detecting if position of tool holder fall down
X46	P7_CN16	Input	Activate M01 code(exact stop)
X47	P8_CN16	Input	Detecting if overload of fan of spindle
M61	P19_CN10	Output	Spindle orientation
M63	P7_CN10	Output	Rotate CW of tool magazine
M65	P20_CN10	Output	Yellow indicator lamp (Stop)
M67	P8_CN10	Output	Red indicator lamp (Fault)
M69	P21_CN10	Output	Green indicator lamp (Running)
M71	P9_CN10	Output	Rotate CCW of tool magazine

M73	P22_CN10	Output	Forward of tool magazine
M59	P6_CN10	Output	Huff/Blower
M32	P9_CN3	Output	Lubrication
M10	P21_CN3	Output	Spindle chuck loose tool
M08	P8_CN3	Output	Coolant
M05	P20_CN3	Output	Stop spindle/Reset M03&M04
M04	P7_CN3	Output	Rotate CCW of Spindle
M03	P19_CN3	Output	Rotate CW of Spindle
M75	P12_CN3	Output	Shift control mode(Position/Speed) of spindle system
Y24	P11_CN16	Output	Arm take/exchange tool, & return to original position
Y26	P13_CN16	Output	Lift tool holder up
Y27	P14_CN16	Output	Fall tool holder down
Y28	P15_CN16	Output	M203: CW_2nd Spindle
Y29	P16_CN16	Output	M204: CCW_2nd Spindle
Y30	P17_CN16	Output	SPEN: Output Enable of SP servo driver
Y31	P18_CN16	Output	Output Enable of 2nd SP servo driver

Note: 1. Tool magazine must backward at position when turn on spindle, also T07 is valid.

2. When M12 is as input point for detecting safe door,it needs to set P7=1 on Other parameter.

3. When M16 is as input point for control chuck loose tool, it needs to set P22=1 on Other parameter.

4. When tool holder fall down(Y27), Position of tool holder must be at position(X41).

4.3.2 Parameters for Arm Tool Magazine

(a) Tool parameter for arm tool magazine

P10=1, Activate the program of tool change;

P11=0, Only Z-axis move to position of tool change;

P12=2, Feeding axes move to position of No.2 reference point when changing tool;

P13=1, Detect SP-orientation end (M22) and Feeding axis move to reference point position;

P14=1, Spindle do orientation when changing tool (M61);

P15=0, Type of tool magazine is arm type;

P16=0, Standard type tool magazine;process of tool change is as chapter4.2.5.

P18=2000, Lifting speed of Z axis when changing tool (unit:mm/min);

P19=5, Delay time after detected T05 when chuck tighten tool (unit: 0.1 sec) ;

P20=5, delay time after spindle loosen tool,also delay time after detected T02(unit: 0.1 sec);

P21=5, delay time after after tool case lifting (unit: ms);

P22=0, Interlock between motion of Z-axis and arm [1:Yes, 0:No], set to "1" means detect X42,position of arm return to home when Z-axis needs to move;

P24=0,Maximum number of tool at area of fixed tool position [after set the parameter, please initialize table of tool case];

P25=1007 , X07 is input point that signal of tool count of magazine,the effective level is rising along [Rising along:"1000+ number", Falling along:"2000+ number"];

P26=1041, X41 is the input point as detect positioning signal of tool magazine[1000+ number];

P27=1001, Y01 is the output point that be used to output control the positive rotation (CW) of tool magazine [1000+ number];

P28=1005, Y05 is the output point that be used to output control the negative rotation (CCW) of

tool magazine [1000+ number];

P29=1042 , X42 is input point that braking point of arm [1000+ number]

P30=1024 , Y24 is output point that control rotation of arm [1000+ number]

P32, Mode of selecting tool/Signal of counting tool[0:Unidirectional; 1: Bidirectional].

P100=0, reference point 1 X (mm) ;

P101=0, reference point 1 Y (mm) ;

P102=0, reference point 1 Z (mm) ;

P103=0, reference point 1 A (mm) ;

P104=0, reference point 2 X (mm) ;

P105=0, reference point 2 Y (mm) ;

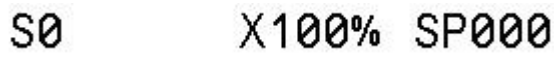
P106=50, reference point 2 Z (mm) ;

P107=0, reference point 2 A (mm) ;

(b) Axis Parameter for offset of spindle orientation

1) P400=1, detect position feedback of spindle when changing tool, set to "0": no detect ;

2) P401, Detecting degree when spindle orientation, manually press K1,after finishing spindle orientation, input value of SP into P401; Eg: current value of SP is 000 as following picture show:



3) P402, Allow error of detecting degree when spindle orientation;

Note: encoder signal of spindle must be connected to system(CN9 plug).

4.3.3 Debug of Arm Tool Magazine

1) Operation of Manual Buttons

K1: Spindle orientation (Indicator of K1 is on after orientation end): Output M61, detect M22 ;

K2: Z-axis move to point of tool changing ;

K3: Setting number of current tool holder;

"**TL-CW**":Magazine rotate CW, Output M63,Pin7_CN10 Plug,detect input of T08& X41(position of tool holder) ;X44 must be valid,also tool holder is up.

"**TL-CCW**":Magazine rotate CCW,Output M71,Pin9_CN10 Plug,detect input of T08 & X41(position of tool holder) ;X44 must be valid,also tool holder is lift up.

2) M Instruction

M71: Fall tool holder down, also reset Y26, output Y27,Pin22_CN10 Plug;& X41 must be valid,also tool holder is at position that can do down/up.

M73: Lift tool holder up, also reset Y27, output Y26;

M65: Arm does one action unconditionally,output Y24. (for debugging)

M881: Spindle Orientation , same function as "K1 key";

3) Instruction of Tool Change

M06 : Change the tool on current tool-case to spindle.

Txx : Rotating Txx to current tool-case; Eg: ***T02 , also rotating No.2 tool to current tool-case.***

M06 Txx: Change tool on current tool-case to spindle ,and then rotate Txx to current tool-case, which prepare for tool changing next time (First change tool and then select tool).

M106 Txx: Move Txx to current tool-case, and then change tool on current tool-case to spindle. (First select tool and then exchange tool,also change Txx to spindle)

M36 : Change Txx to spindle in single step mode. After finish one step,system will be suspended, Press “Start”button again to execute the next step(for debugging).

5. Process of Tool Changing

① Z axis move to position of tool changing (Z axis must return to home firstly);

② Spindle orientation: output M61, detect M22(position of backing zero) ;

③ Tool case falling: output Y27, detect X45(falling in position), and X41(position of tool magazine) must be in place;

④ Arm hold tool in SP: output Y24, detect X42(position of arm hold tool);

⑤ SP loose tool: output M10, detection T02(position of spindle loose tool);

⑥ Arm take tool and change tool:output Y24,detect X42(position of arm take tool);

⑦ Spindle tighten tool: reset M10, detect T05 (position of spindle tightening tool);

⑧ Arm return home: output Y24, detect X42(position of arm return home);

⑨ Lifting tool case: output Y26, detect X44(position of lifting).

Special attention:

1. When power off suddenly or emergency stop was happening during changing tool,which cause the tool case is in a mess , please rotate several tool position of magazine manually and then use K3 to set the number of current tool case;

2. For this kind of tool magazine, we could set the No.24 parameter in tool parameter ,which is set for Maximum number of tool at area of fixed tool position, it's better to initialize the table of tool case tafter setting the parameter.

Example: Set P24=8 ,which means that each tool case from no.1 to no.8 only put into related tool by one-one. It isn’t random that select No.1~No.8 tool, which is fixed installation. This function could be used to installing tool magazine which is large diameter cutter;

3. The position of No 0 tool (T0) should not have tool, otherwise perhaps occur an accident;

4. M65 doesn't check the condition, only stop when emergency stop, pay more attention when using, otherwise perhaps occurs accident.

4.4 Special Umbrella Type Tool Magazine

1. Tool parameter:P15=2, P16=16, type of tool magazine is set to "Special Umbrella Magazine ".

2. Editing control program of tool changing,“ProgramTool”, and restore “ProgramTool” into system. Running “ProgramTool” program from 3rd step of process of standard type umbrella type tool magazine. The other operations are same to operations of standard umbrella type tool magazine.

4.5 Special Arm Type Tool Magazine

1. Tool parameter: P15=0, P16=64, type of tool magazine is set to "Special Arm Magazine"; Tool parameter: P29=0, P30=0.

2. Editing control program of tool changing,”ProgramM6”,and restore “ProgramTool” into system. Programming of “ProgramM6” begin from 3rd steps of process of standard arm type tool magazine, the other operations are same to operations of standard arm type tool magazine.

4.6 Linear Type Tool Magazine

1. Tool parameter: P15=1,P16=128,type of tool magazine is set to "Linear Magazine";

2. Editing control program of tool changing, “ProgramTool”,and restore “ProgramTool” into system. All process of tool changing are finished by control program- “ProgramTool”.

4.7 Special Servo Type Tool Magazine

1. Tool parameter: P15=0,P16=32,type of magazine is set to "standard Arm Servo Magazine";
or P16=32+64=96,set to “Special Arm Servo Magazine”.

2.Tool Parameter:P15=2,P16=32,,type of magazine is set to "standard Umbrella Servo Magazine";
or P16=32+16=48,set to “Special Umbrella Servo Magazine”.

70,Servo tools go home speed(degree/min): Speed of servo tool magazine go home in positive direction,unit is degree/min.

71,Servo tools go home reverse speed(degree/min):Speed of servo tool magazine go home in negative direction,unit is degree/min.

72,Servo tools go home offset(0.001degree): Offset of servo magazine after homing.

73,Servo change tool speed(degree/min): Rotating speed when servo magazine select tool.

74,Servo tools axis(1=A; 2=B; 3=C): Axis that is used for tool changing in servo magazine.

75,Acceleration of servo magazine moving, unit is degree/min/s.

76,Max distance to detect zero position of encoder when magazine homing(0.001degree)

77,Detect zero position of encoder when servo magazine homing (0: is No detect. 1: Yes)

78,Pulses that servo magazine rotate from current tool to next tool.

79,Current tool number after servo magazine homing.

2. When special type arm servo type tool magazine(P16=96), Editing control program of tool changing, “ProgramM6”,and restore “ProgramM6” into system. Programming of “ProgramM6” begin from 3rd steps of process of standard arm type tool magazine, the other operations are same to operations of standard arm type tool magazine.

3. Home of Servo Magazine: P74=1,detect A0 signal; P74=2,detect B0 signal; P74=3,detect C0 signal; P77=1,detect zero position of motor’s encoder.

4. Output signal Y25,lock/loose of servo magazine, Input point: X43, detecting Position Signal.If no need lock/loose signal,short connected Y25 to X43.

4.8 Taiwan H.D.W. Disk Arm Type Tool Magazine

1. Tool parameter

P16=64,set to "special arm type"; tool parameter P29=0 P30=0.

2. Edit of Control Program

Editing control program of tool changing,”ProgramM6”,and restore “ProgramTool” into system. Programming of “ProgramM6” begin from 3rd steps of process of standard arm type tool magazine, the other operations are same to operations of standard arm type tool magazine.

3. M instruction

M71: Falling of tool case, reset Y26, output Y27, tool case locating check X41 must be valid;

M73: Lifting of tool case , reset Y27, output Y26;

OUT+Y30: Magazine rotation one step unconditionally, for debugging,output Y30;

M881: Same function as "K1 key";

4. Table of IO Ports

I/O	Signal	Arm Type Tool Magazine	Disk Arm type magazine
X00	T01	Detecting if overload of magazine's motor	
X01	T02	Detecting if position of Chuck loosing tool	
X02	T03	Detecting if level of lubricating oil is low	
X03	T04	Detecting if coolant water is lacking	
X04	T05	Detecting if position of Chuck tightening tool	
X05	T06		Signal of clasping tool S6（Right）
X06	T07		Signal of original point S7（Left）
X07	T08	Counting of tools	Count of Tool magazine S1
X23	ALM1	Alarm input of Spindle Driver	
X24	ALM2	Detecting if overload of cooling motor	
X27	M22	Detecting if spindle orientation end	
X29	M12	Detecting if safe door of machine tool is open	
X30	M14	Detection for pressure alarm of compressed air	
X31	M16	External Switch for SP loosen tool	
X40		Detecting if overload of motor for removing chips	
X41		Detecting if position of tool holder	Position of tool magazine S2
X42		Detecting if position of arm(take/exchange/return)	Confirm signal of braking S5（Middle）
X44		Detecting if position of tool holder lift up	Position of returning tool S4
X45		Detecting if position of tool holder fall down	Position of tool fallen S3
X46		Activate M01 code(exact stop)	
X47		Detecting if overload of fan of spindle	
Y00	M61	Spindle orientation	
Y01	M63	Rotate CW of tool magazine	Output Rotation CW of Motor
Y02	M65	Yellow indicator lamp (Stop)	
Y03	M67	Red indicator lamp (Fault)	
Y04	M69	Green indicator lamp (Running)	
Y05	M71		Output Rotation CCW of Motor
Y07	M59	Huff/Blower	
Y08	M32	Lubrication	
Y10	M10	Spindle chuck loose tool	
Y11	M08	Coolant	

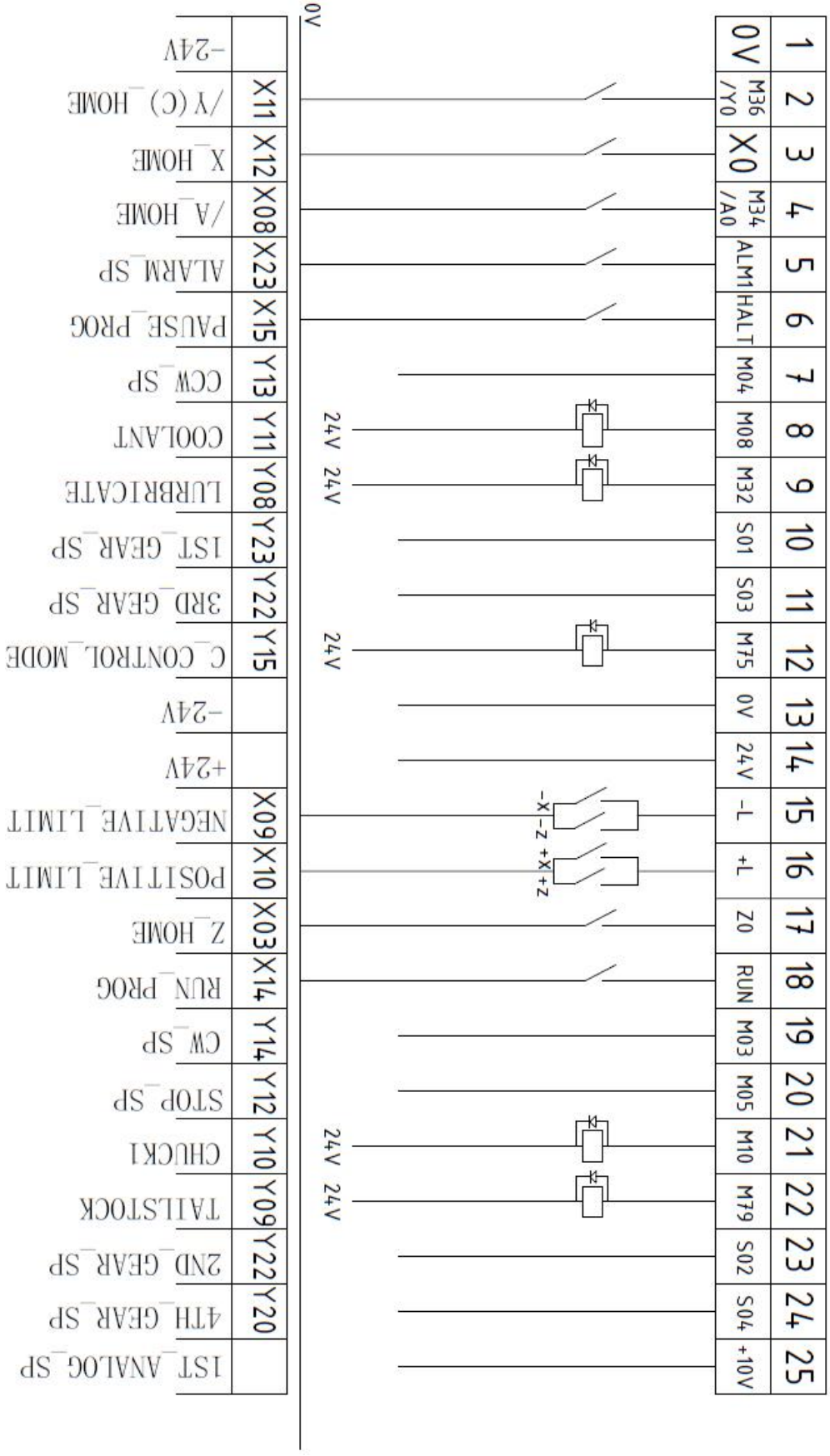
Y12	M05	Stop spindle/Reset M03&M04	
Y13	M04	Rotate CCW of Spindle	
Y14	M03	Rotate CW of Spindle	
Y15	M75	Shift control mode(Position/Speed) of spindle system	
Y26		Lift tool holder up	Solenoid valve of tool returning S10
Y27		Fall tool holder down	Solenoid valve of tool fallen S8
Y28			Output Enable of servo driver of feeding axis
Y29		SPEN: Output Enable of SP servo driver	Output Enable of SP servo driver
Y30		Arm take/exchange tool, & return to original position	Motor of the tool magazine structure
Y31		SPEN: Output Enable of 2nd SP servo driver	

Note:1.Make reversed of feedback signals from tool magazine by IC2803,shown as Fig.4.8.1, also transfer PNP type signals to NPN type signal,which can be acceptable by CNC system.

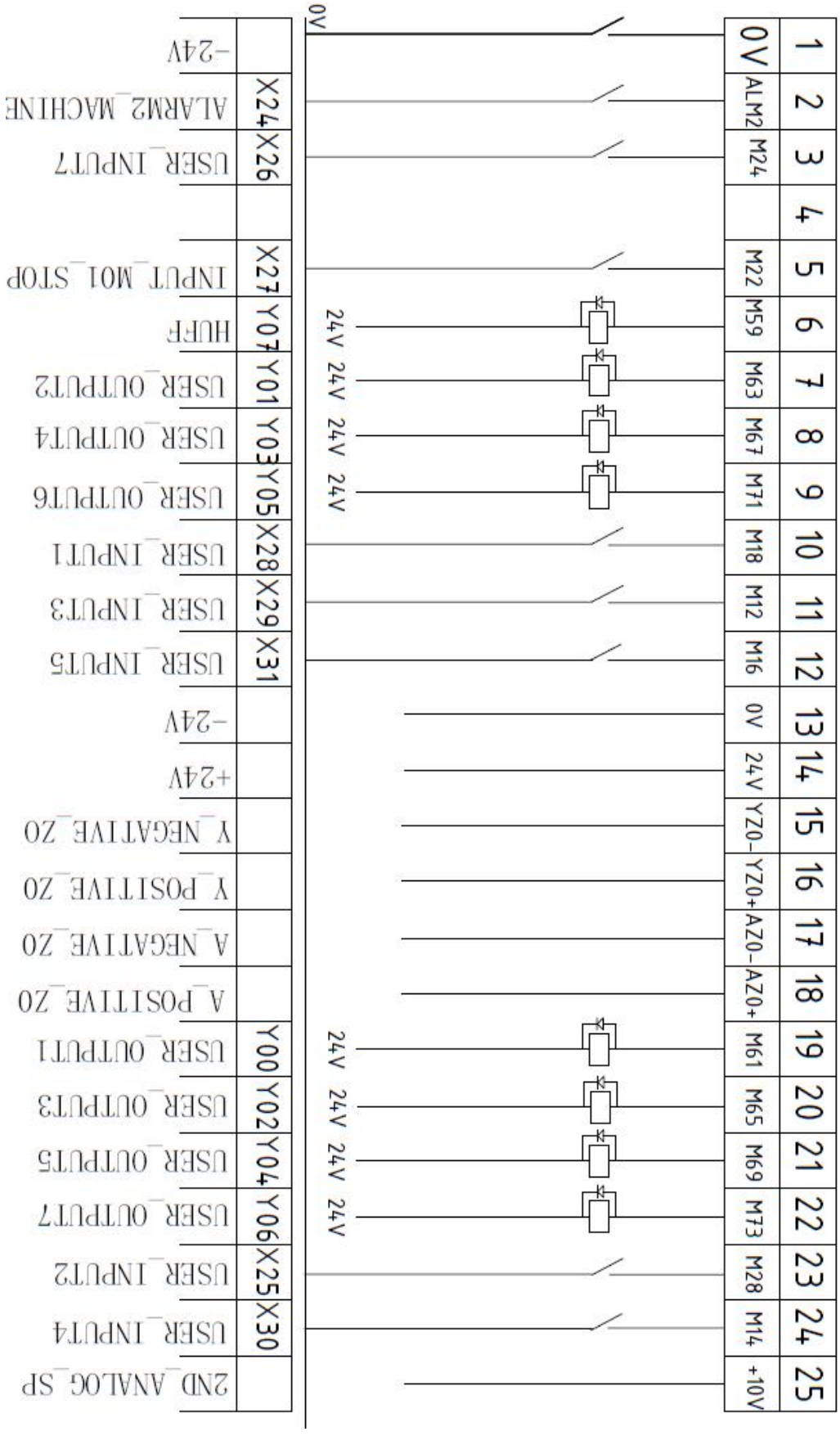
2.Valid level of all inputs & outputs is 0V.

Fig.4.8.1 Transformer board_2803IC

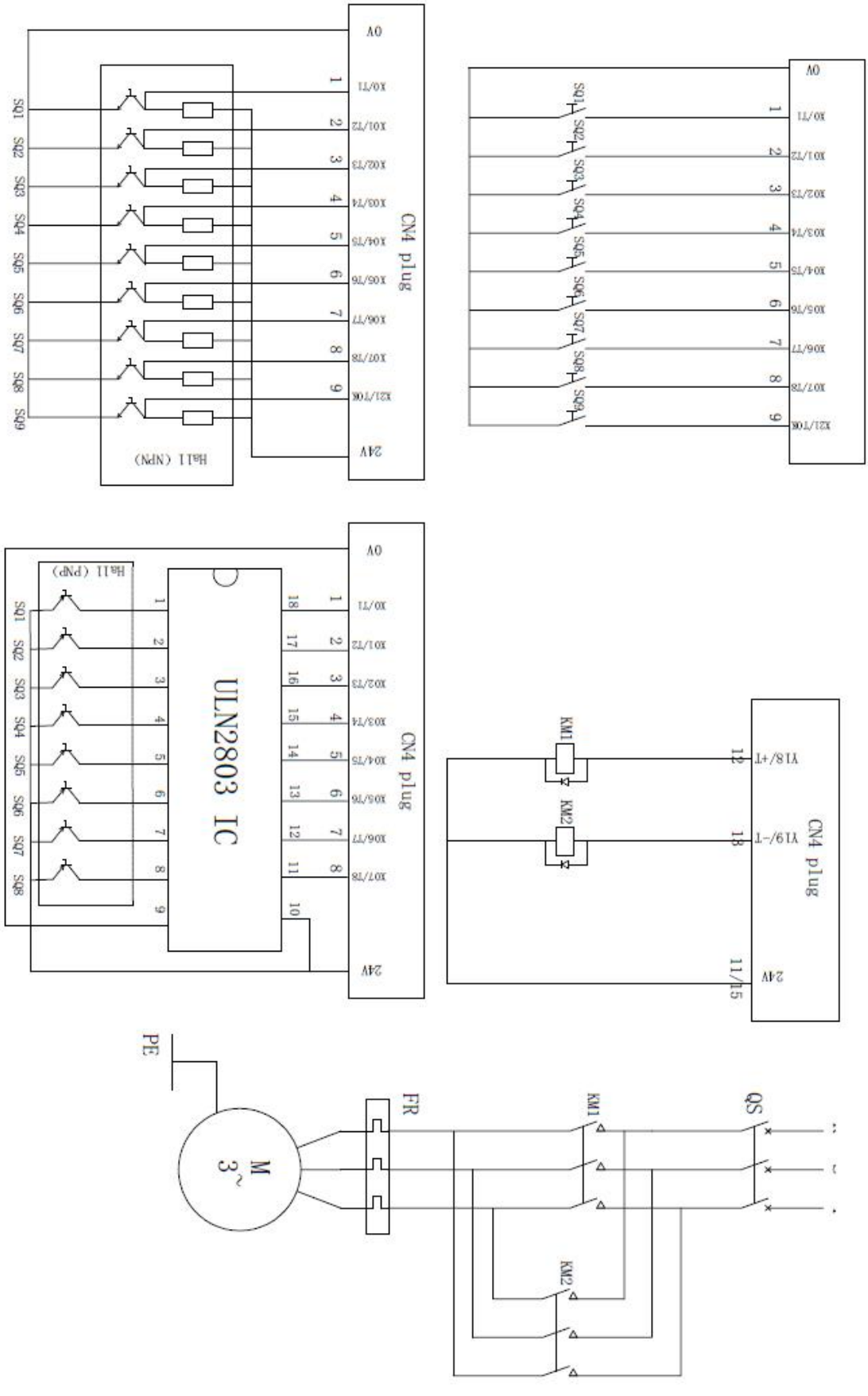
Appendix I: Wiring Diagram of CN3 Plug



Appendix II: Wiring Diagram of CN10 Plug



Appendix III: Wiring Diagram of CN4 Plug



Appendix IV Operational Panel

A Type Operational Panel

B Type Operational Panel

C Type Operational Panel(Default Configuration)

E Type Operational Panel

Note:SZGH-CNC1000MDcb series CNC controller can be configured with any type operational panel.

Appendix V:Wiring Diagram of Bamboo Hat Type Tool Magazine

Configure with SZGH brand spindle servo system(V004)

A. Table of Wiring(Controller To Spindle Servo Driver)

CN3_CNC	PIN	Connect	PIN	X5A_SP	Remark
GND	13	⟷	23	OCOM,AS-	Com port of Outputs is 0V
		⟷	26	AS-	Ground_Analog
24V	14	⟷	22	SEL	Supply power +24V
M03	19	⟷	10	I0	CW Rotation of SP
M04	7	⟷	19	I1	CCW Rotation of SP
+10V	25	⟷	25	AS+	Analog 0~10V(Speed mode)
ALM1	5	⟷	14	O0	Alarm of Spindle
M75	12	⟷	21	I4	Set of Control Mode
CN10_CNC	**PIN**	**Connect**	**PIN**	**CN1_SP**	**Remark**
M61	19	⟷	1	I2	Output Spindle Orientation
M22	5	⟷	24	O1	Detection SP Orientation
CN10_CNC	**PIN**	**Connect**	**PIN**	**CN3_SP**	**Remark**
SPCP+	19	⟷	2	EP+	Positive Pulse Signal of SP
SPCP-	7	⟷	11	EP-	Negative Pulse Signal of SP
SPDIR+	25	⟷	4	ED+	Positive Direction Signal of SP
SPDIR-	5	⟷	13	ED-	Negative Direction Signal of SP
CN9_CNC	**PIN**	**Connect**	**PIN**	**CN3_SP**	**Remark**
PA+	5	⟷	OA+	9	A+ Phase signal of Encoder
PA-	7	⟷	OA-	18	A- Phase signal of Encoder
PB+	3	⟷	OB+	17	B+ Phase signal of Encoder
PB-	6	⟷	OB-	8	B- Phase signal of Encoder
PC+	2	⟷	OZ+	7	Z+ Phase signal of Encoder
PC-	8	⟷	OZ-	16	Z- Phase signal of Encoder

Note: The default resolution of SP_encoder is 1024PPR.

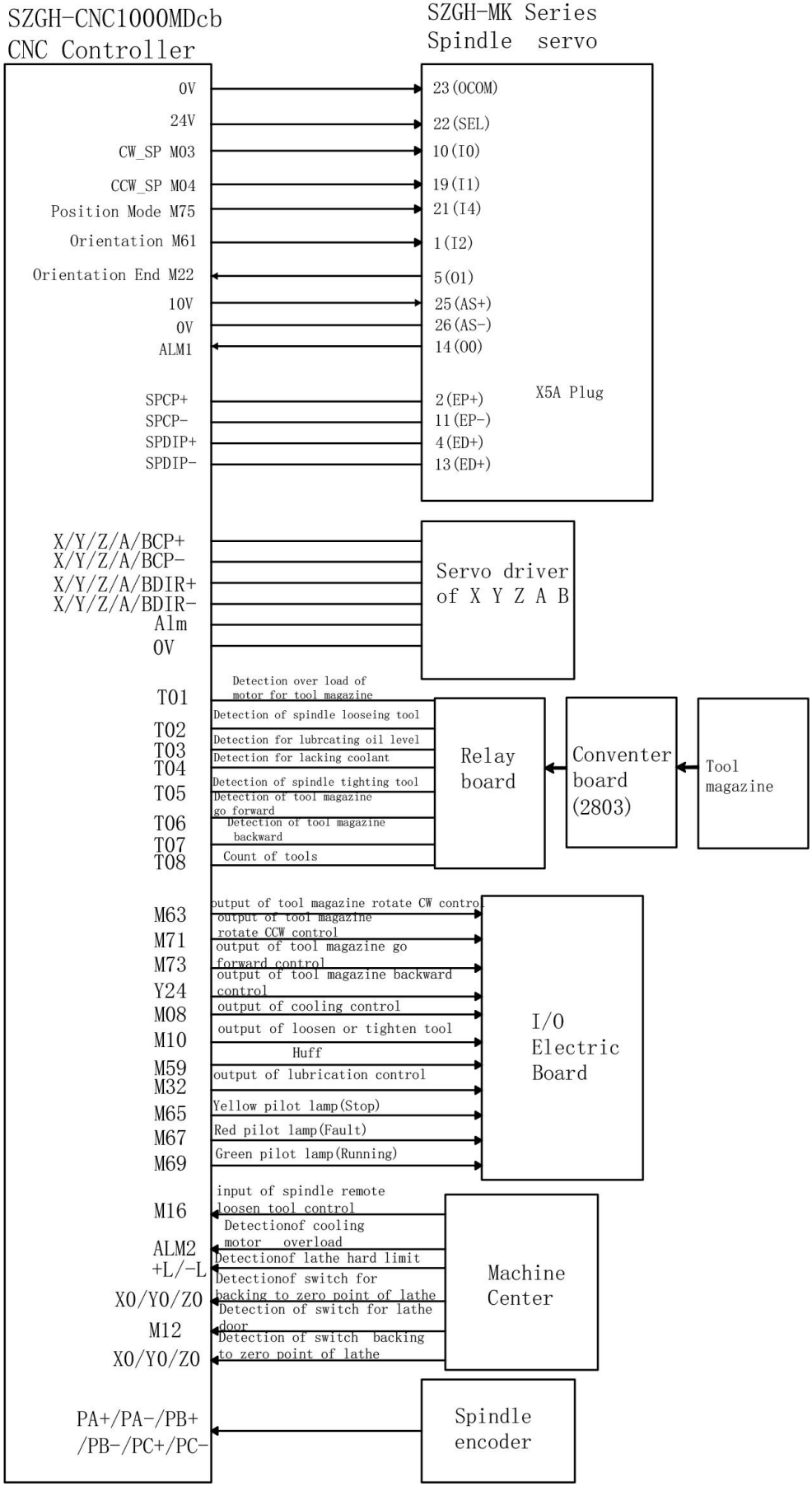
B. Set of Parameter

P22_01: set to 3, input pulse of position mode is pulse+direction.

M75: Output Y15, set control mode to Position control mode;

M03/M04/M74: Reset Y15, set control mode to Speed control mode.

C: Wiring Diagram(SZGH-CNC1000MDcb series & Spindle Servo System_V004)



Системы ЧПУ:

SZGH-CNC1000MDi & CNC1000TDi
SZGH -GH1000MC
SZGH -GH1000TC
SZGH-CNC990MDb-3/4
SZGH-CNC990MDc-3
SZGH-CNC1000MDb-3/4/5
SZGH-CNC1000MDc-3
SZGH-CNC1000GDb-2/3/4/5
SZGH-CNC1000TDc-2/3/4/5

Шпиндельные приводы и серводрайверы:

SZGH08-3-9.5-1.5/2.2-4-1500 - SZGH-S4T1P5
SZGH08-4-14-2.2/3.7-4-1500 - SZGH-S4T2P2
SZGH09-2-24-3.7/5.5-4-1500 - SZGH-S4T5P5
SZGH10-2-35-5.5/7.5-4-1500 - SZGH-S4T7P5
SZGH10-3-48-7.5/11-4-1500 - SZGH-S4T011
SZGH10-4-71-11/15-4-1500 - SZGH-S4T015
SZGH13-2-95-15/18.5-4-1500 - SZGH-S4T018
SZGH13-3-117-18.5/22-4-1500 - SZGH-S4T022
SZGH13-4-140-22/30-4-1500 - SZGH-S4T030
SZGH16-1-191-30/37-4-1500 - SZGH-S4T037
SZGH16-2-235-37/45-4-1500 - SZGH-S4T045
SZGH16-3-286-45/55-4-1500 - SZGH-S4T055
SZGH16-4-350-55/75-4-1500 - SZGH-S4T075
SZGH18-1-478-75/90-4-1500 - SZGH-S4T090
SZGH18-2-573-90/110-4-1500 - SZGH-S4T110
SZGH18-3-700-110/132-4-1500 - SZGH-S4T132
SZGH22-1-840-132/160-4-1500 - SZGH-S4T090