

User Manual

SZGH-CNC1000MDi(series)

Milling Control System

-OPERATION-

V3.2

-BOOK2-

Shenzhen Guan hong 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

SAFETY PRECAUTIONS.....	II
Chapter 1 Operation.....	1
1.1 Operational Panel.....	1
1.2 Function Menu.....	1
1.3 Editing Keyboard.....	2
1.4 Machine Control Panel.....	2
1.5 Manual Operation.....	5
1.5.1 Manual Continuous.....	5
1.5.2 Manual Increment.....	6
1.5.3 Manual pulse generator(Handwheel).....	6
1.5.4 Reference Point/Home Return.....	7
1.6 Auto Operation.....	9
1.6.1 Automatic Processing Mode.....	9
1.6.2 Processing at arbitrary program line or with arbitrary tool.....	10
1.6.2.1 Start from "nth" real line(block).....	10
1.6.2.2 Start from "N**" line.....	10
1.6.3 Start Program.....	11
1.6.4 Halt Program.....	11
1.6.5 Emergency Stop.....	11
1.6.6 Alarm.....	11
1.6.7 Indicator Light Output.....	12
1.6.8 DNC function.....	12
1.6.8.1 RS232-DNC.....	12
1.6.8.2 USB-DNC.....	13
1.7 External Electrical Connection.....	13
1.7.1 Limitation.....	13
1.7.1.1 Software limitation.....	13
1.7.1.2 External Switch for limitation.....	14
1.7.1.3 Suggestion Usage.....	14
1.7.1.4 Power ON/OFF CNC.....	15
1.8 Diagnosis.....	16
1.9 Programming Operation.....	19
1.9.1 Editing.....	19

1.9.2 Copy.....	21
1.9.3 Delete.....	21
1.9.4 Rename.....	21
1.9.5 Information.....	21
1.9.6 Compile.....	22
1.9.7 Folder management.....	22
1.9.8 Execute Program.....	22
1.9.9 Communication.....	22
1.9.10 U-disk management.....	22
1.9.10.1 Function Keys of USB-disk.....	23
1.9.10.2 Management of Processing Program.....	23
1.9.10.3 Management of Parameters & Software.....	24
Chapter 2 Parameter List.....	25
2.1 User Parameter.....	26
2.2 Speed parameter.....	32
2.3 Axis parameter.....	42
2.4 Tool parameter.....	54
2.5 Other Parameter.....	56
2.6 Workpiece Coordinate Parameter.....	71
2.6.1 How to set up the workpiece coordinate system?.....	72
2.6.2 How to adjust the offset value for workpiece coordinate system after set well?.....	73
2.7 Password.....	74
2.8 Redeem.....	75
2.8.1 Radius Compensation.....	75
2.8.2 Length of redeem.....	76
2.8.3 Tool Sets List.....	77
2.8.4 Set quantity.....	77
2.9 Screw Compensation.....	78
Chapter 3 Installation & Connection.....	81
3.1 System Installation.....	81
3.2 System installation dimension.....	81
3.3 System Rear View.....	82
3.4 Interface Connection Graph.....	83
3.4.1 Communication Socket (Female/DB9).....	83

3.4.2 Spindle Encoder Socket (Female/DB9).....	85
3.4.3 CN11 MPG/Handhold Box Socket (Male/DB26).....	86
3.4.4 CN21 Pulse Signal Socket (Female/DB26).....	88
3.4.5 Debug for normal absolute type servo motor.....	89
3.4.6 CN22 Modbus Communication Socket(Female/DB15).....	90
3.4.7 IO Ports on IO board.....	91
3.5 Daily Maintenance and Repair.....	95
3.5.1 Maintain.....	95
3.5.2 Ordinary Problem.....	95
Chapter 4 CNC Machining Center.....	96
4.1 Status of CNC Machine Center System.....	96
4.2 Umbrella Tool Magazine.....	96
4.2.1 I/O ports for Umbrella Tool Magazine.....	96
4.2.2 Parameters for Umbrella Tool Magazine.....	97
4.2.3 Debug of Umbrella Tool Magazine.....	98
4.3 Standard Arm Type Tool Magazine.....	99
4.3.1 I/O ports for Arm type Tool Magazine.....	99
4.3.2 Parameters for Arm Tool Magazine.....	100
4.3.3 Debug of Arm Tool Magazine.....	101
4.4 Special Umbrella Type Tool Magazine.....	102
4.5 Special Arm Type Tool Magazine.....	102
4.6 Linear Type Tool Magazine.....	103
4.7 Special Servo Type Tool Magazine.....	103
4.8 Taiwan H.D.W. Disk Arm Type Tool Magazine.....	103
Appendix I: Wiring Diagram of CN3 Plug.....	106
Appendix II: Wiring Diagram of CN10 Plug.....	107
Appendix III: Wiring Diagram of CN4 Plug.....	108
Appendix IV:Wiring Diagram of Bamboo Hat Type Tool Magazine.....	109

Chapter 1 Operation

When using modbus series CNC Milling system, just master the parameters list of system, program edit, 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)

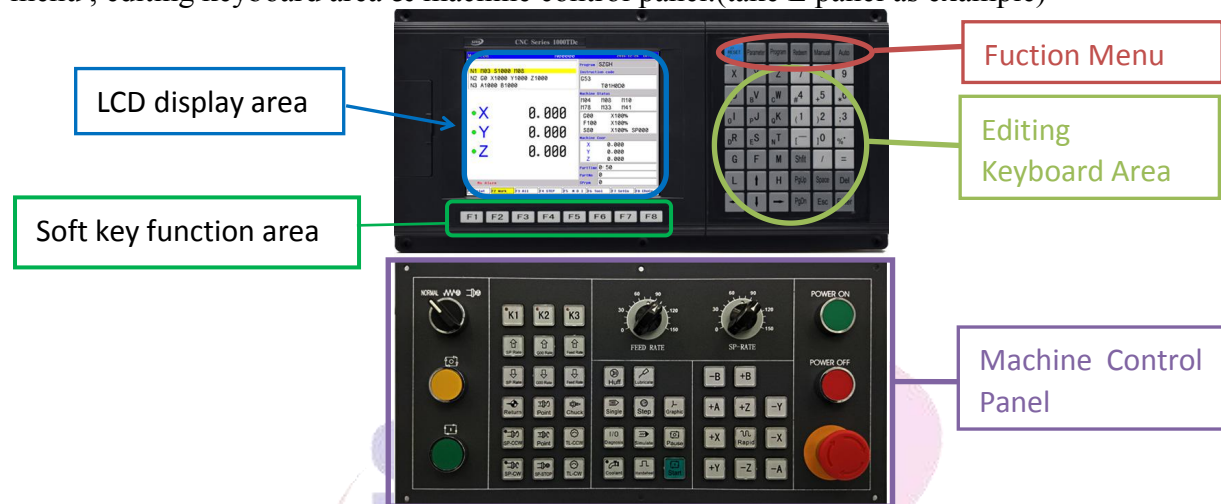
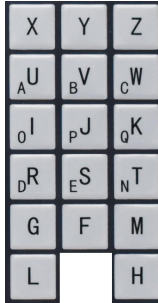
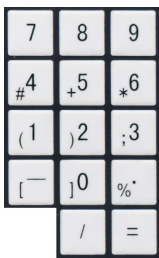
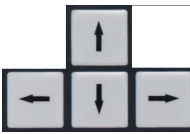
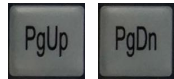


Fig1.1 SZGH-CNC1000MDi CNC Milling Controller

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	<i>Note: Clear number of workpiece</i>
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 & rapid function for set manual feeding speed with number, 0-9.	
	Cursor Move Keys	Move the cursor in different directions.	
	Page up/down key	Page up/down on display	<i>Note: Exchange of coordinate</i>
	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	Chuck switch	Tighten/Loosen Tool of Spindle	Control code: M10/M11 Output Point: PIN21 of CN3 plug
 Lubricate	Lubricate switch	Tailstock Forward/Backward	Control code: M32/M33 Output Point: PIN9 of CN3 plug
 Huff	Huff switch	Huff ON/OFF	Control code: M59/M58 Output Point: PIN6 of CN10 plug
 Single	Single block key	To enter single block mode	
 Step	Per Step mode key	To enter single step mode	Switching cycle from "manual continuous" to "manual increment"
 Graphic	Graphic mode key	To enter graphic mode(Only Work on AUTO condition)	
 Simulate	Simulate switch/dry run switch		
 Handwheel	Handwheel/*1/*10/*100	Enter controlling condition of handwheel. When P1=0 on Other parameter, press Handwheel repeatedly, it will shift handwheel rate of *1/*10/*100	
 Diagnosis	Diagnosis key	Enter the interfaces of diagnosis	
 Pause	Pause key	Halt for program	
 Start	Cycle start key	Press this key and the system mode automatically runs	Auto mode, MDI mode, DNC mode
 Return	Return Home of Machine key	To enter mode of return home of machine	

Key	Designation	Explanation	Remarks and operation explanation
 SP Rate	Spindle override keys	Up/Down rate of SP_Speed	Range is from 5% to 150%,16 gears totally,only work for 1 st Spindle (P2= 0 on Axis)
 SP Rate			
 G00 Rate	Rate of rapid speed	Up/Down rate of Rapid_Speed	range is from 5% to 100%,6 gears totally
 G00 Rate			
 Feed Rate	Rate of feeding speed	Up/Down rate of Feeding Speed	range is from 0% to 150%,16 gears totally (P1=0 on Axis parameter)
 Feed Rate			
 SP-CW	Spindle control keys	CW/CCW/Stop of Spindle	Corresponding to M03/M04/M05 codes
 SP-CCW			
 SP-STOP	Rotation of SP in JOG	Point-JOG CW/CCW rotation of spindle	
 Point			
 Point			

	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 <i>Note:only E type operational panel has Pause button.</i>	
	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 manual & auto 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. <i>Note: only A type & C type operational panel have handwheel(MPG).</i>	
	Power On/Power OFF switches <i>Note:Only C type & E type operational panel have these switches.</i> <i>Using solution , please check 1.7.1.4_book2</i>	

Note: 1."Manual speed controlled": Press "0" "1" "2" "3" "4" "5" "6" "7" "8" "9" number keys to set feeding speed with "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.48 parameter in speed parameter.

6."I": Modify the increment per step in manual increment mode(Step key)

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.

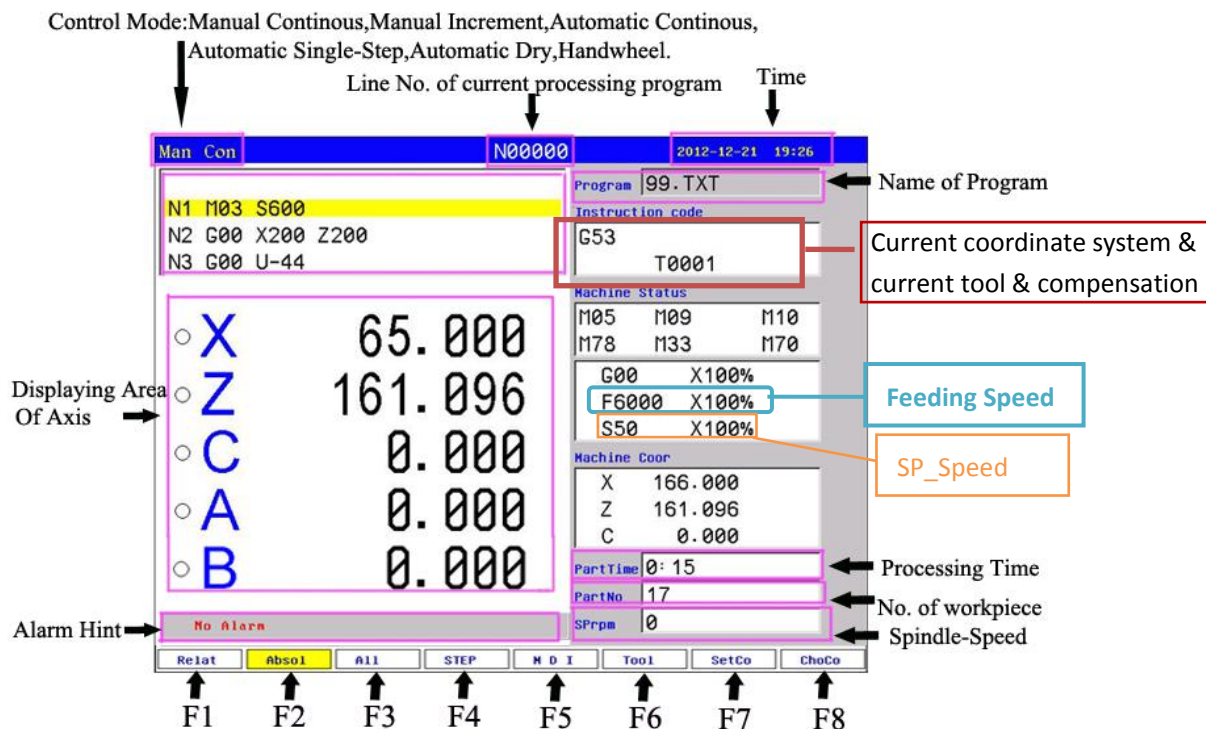


Fig1.2 Displayer

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,user can adjust machine coordinate , auxiliary actions & operation, posit tool , starting point of processing workpiece,and process workpiece manually.

1.5 Manual Operation

1.5.1 Manual Continuous

Continuous operation is basis on the time of pressing, Press down to feed, up to stop feeding. Making sure the axis and using "Manual Feeding Key"(also +X, -X, +Y,-Y, +Z,-Z, +A,-A, +B,-B) to feed, feeding speed displays on the interface (F) multiply Feed Rate.

When continuous starting, press "Rapid" to switch the speed set by P1,P2,P3,P4,P5 in Speed parameter, also G00 speed. If manual feed speed is higher than P1-P5 speed parameters,feed speed will be P1,P2,P3,P4,P5 in Speed parameter times rapid override(also G00 Rate).

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 value.

When feed is over max travel of axes,(software limit or hardware limit point), stop feed and only prompt to feed with reverse direction.

Maximum manual feed speed is limited by P6 in Speed parameter, when setting the speed is higher than the value of parameter, then will be P6 in Speed parameter.

When P38=8 in other parameter, "Rapid" is change into a repeatable switch, press once to shift to Rapid mode(blink display "M-Fast" on screen), press again to shift back to normal mode.

1.5.2 Manual Increment

Press "Step" key to enter Manual Increment mode, and then set manual incremental value by press "I" key, which will hint "Input Manual Increase Value(mm):" ,and input value & press Enter to confirm. press "Manual Feeding Key"(also +X, -X, +Y,-Y, +Z,-Z, +A,-A, +B,-B) once , CNC will run a value of increment. Default display "Man Inc 1.000" ,means CNC is in manual increment mode with the increment value is 1mm, press "I" to revise increment value and Enter.

Its 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, which is set by P1 on Other parameter.

Handwheel in operate panel(A type or C type):(P1=0 on Other parameter), Press "Handwheel" key to enter "handwheel" mode, press alphabet keys "X,Y,Z,A,B" to select feeding axis, press "Handwheel" key again to shift Handwheel rate.

Handwheel in Handhold Box: (P1=1 on Other parameter) Press "Handwheel" key to enter handwheel mode. User can operate the axes selection & feeding rate of handwheel(*1/*10/*100).

Handwheel is mainly used for "Tool", also posit tool. And its speed of handwheel is decided by increment per step & speed of rotating handwheel, normally lower than 200r/min(100 pulses per cycle).

In Speed parameter:

P26 is set for acceleration / deceleration time constant.

P27 is set for acceleration / deceleration time constant when simulate.

P28 is set for G00 speed when use Handwheel to run cnc

P29 is set max speed of X-axis with handwheel.

P30 is set max speed of Y-axis with handwheel.

P31 is set max speed of Z-axis with handwheel.

P32 is set max speed of A-axis with handwheel.

P33 is set max speed of B-axis with handwheel.

Note: 1. SZGH-CNC1000MDi series support handwheel both in panel & in handhold box.

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



Fig1.3 Handhold Box(Manual Pulse Generator)

1.5.4 Reference Point/Home Return

The CNC machine tool has a position, where is used to determine the machine position. This position is called the reference position, also machine zero point, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on or alarm/release emergency stop, the tool need moved to the reference position again for setup right machine coordinate system.

Manual reference position return is to move the tool to the reference position by push Return buttons on the operate panel. After home done, which will clear machine coordinate value to 0.

P45 in Axis parameter 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 (Suggest if absolute servo)

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,

P39: Bit parameter, D3:X ; D4:Y ; D5:Z ; D6:A ; D7:B ; D8: C ; D9: XS ; D10:YS ; 1 means float zero point;

P40: Distance between reference position & current position in X direction;

P41: Distance between reference position & current position in Y direction;

P42: Distance between reference position & current position in Z direction;

P43: Distance between reference position & current position in A direction;

P44: Distance between reference position & current position in B direction;

P44a: Distance between reference position & current position in C direction;

P44b: Distance between reference position & current position in XS direction;

P44c: Distance between reference position & current position in YS direction;

Steps of setting floating zero point(home):

1. Enabled function of float zero point of all axes(XZCABCXsYs): P23= 0000011111111001;
2. Moving each axes to designated position in order to set floating point easily.
3. Set distance between reference & designated position: P40/P41/P42/P43/P44/P44a/b/c

Note: If current position is home of machine coordinate system, just set P40-P44 to 0 for each axis.

(2) Switch For Homing Set (Suggest if Increment Servo/Stepper)

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

A. Parameters set: in Axis Parameter,

P39: Bit parameter, D3:X ; D4:Y ; D5:Z ; D6:A ; D7:B ; 0 means switch for homing;

P45: Grade of Homing, 0: Just hint; 1: No use; 8: Compulsion; 9: Super Compulsion;

P46: Mode of Homing, 0 & 2: Detect Z0 signal; 1 & the others, no detect ;

P47: Bit Parameter, Direction & Sequence of Homing;

P48: Bit parameter, Type of switch for homing ;

P49: Range of detecting Z0 signal in X-axis ;

P50: Range of detecting Z0 signal in Y-axis ;

P51: Range of detecting Z0 signal in Z-axis ;

P52: Range of detecting Z0 signal in A-axis ;

P53: Range of detecting Z0 signal in B-axis ;

P54: Offset after homing in X-axis

P55: Offset after homing in Y-axis

P56: Offset after homing in Z-axis

P57: Offset after homing in A-axis

P58: Offset after homing in B-axis

P58-1: Offset after homing in C-axis

P58-2: Offset after homing in Xs-axis

P58-3: Offset after homing in Ys-axis

In Speed Parameter,

P38: X-axis homing speed

P40: Y-axis homing speed

P42: Z-axis homing speed

P44: A-axis homing speed

P46: B-axis homing speed

P121: C-axis homing speed

P122: Xs-axis homing speed

P123: Ys-axis homing speed

P39: Speed for off of homing switch & detecting Z0 signal of X-axis

P41: Speed for off of homing switch & detecting Z0 signal of Y-axis

P43: Speed for off of homing switch & detecting Z0 signal of Z-axis

P45: Speed for off of homing switch & detecting Z0 signal of A-axis

P47: Speed for off of homing switch & detecting Z0 signal of B-axis

P47-1: Speed for off of homing switch & detecting Z0 signal of C-axis

P47-2: Speed for off of homing switch & detecting Z0 signal of Xs-axis

P47-3: Speed for off of homing switch & detecting Z0 signal of Ys-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	Address	Function
X0	X12	X axis homing switch input
Y0	X13	Y axis homing switch input
Z0	X11	Z axis homing switch input
A0	X08	A axis homing switch input
M24/B0	X26	B axis homing switch input
M28/C0	X25	C axis homing switch input
XS0	X17	Xs axis homing switch input
YS0	X16	Ys axis homing switch input

Note: 1. when system hasn't B axis & C-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) Operation of Return Reference Position

Press "Return" in Manual mode, system will hint "Input axis name:(X; Y; Z ; A; B; C; XS; YS ; 0(ZXYABCXSYS))", 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 Chapter3.10_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 cause troubles or dangerous.

1.5.5 Setup Workpiece Coordinate System

Steps of setup workpiece coordinate system

1) Press "F8-ChoCo" or "F5_MDI" key to enter MDI mode on Manual Condition

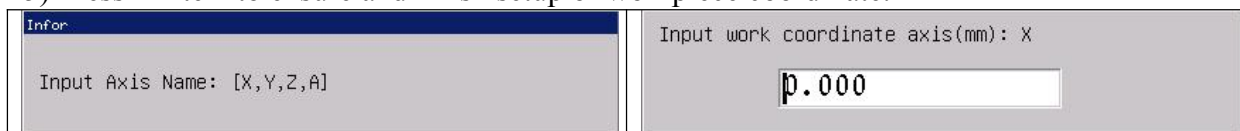


2) Input "G54 ~ G59" or Press "F8 ChoCo" to select workpiece coordinate system

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.



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
● U	0.000	○ X	0.000	Relative	0.000	Machine	0.000	
● V	0.000	○ Y	0.000	U	0.000	X	0.000	
● W	0.000	○ Z	0.000	V	0.000	Y	0.000	
● A	0.000	○ A	0.000	W	0.000	Z	0.000	
				A	0.000	A	0.000	
				Work	0.000	Distance to go	0.000	
				X	0.000	X	0.000	
				Y	0.000	Y	0.000	
				Z	0.000	Z	0.000	
				A	0.000	A	0.000	

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)

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

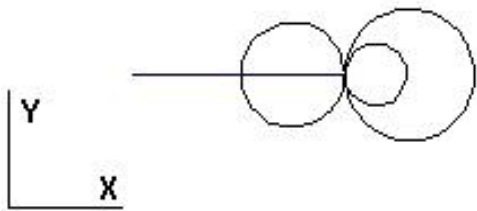
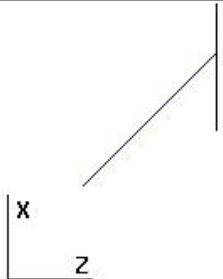
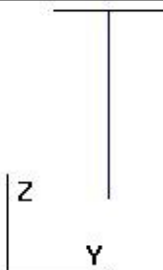
"Coordinate/Graphic": Press "Graphic" in Auto mode to switch cycle.

"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" real line(block)

At the condition of automatic processing, press "—" to pop-up a dialog box, import a number of line, press "Enter" to confirm, or press "↓ (Up/Down arrow key)" to select starting block line, 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 P8 in Speed parameter(Default speed of G01/G02/G03), 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 P8 in Speed parameter, then run the program normally.

1.6.2.3 Start from "Tool" line

The system support that program can start from one Tool. At automatic condition, press "G" to pop up a dialog box to import tool number, press "Enter" to confirm. Press "Start" to run program at the "Tool" line you import(mark).

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

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) 3-Position switch turns to the middle or right side.
- (4) Press "Reset" to stop all the actions of program.
- (5) Fix external switch to Halt port.

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. When set P29 on other parameter to 1, when CNC alarm, which will output M67(Y03).

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(X09) +L(X10)
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(X20/X22)
Spindle driver is alarming	ALM1(X23)
Machine tool is alarming	ALM2(X24)
Emergency Stop	Stop(PIN5_ CN11 Plug)

There are also some user-defined alarm as following

Alarm Hinting Message	Auxiliary Relay in PLC
ALM2:Machine is alarming	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(X20).

P17-1: Type of alarm for servo driver, 0: NO type, 1: NC type(X22).

P18: Type of alarm for spindle, 0: NO type, 1: NC type.(X23)

P19: Type of alarm for machine tool, 0: NO type, 1: NC type.(X24)

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(Y04)	P28=1 In Other parameter
Program Halt	M65(Y02)	
Alarm	M67(Y03)	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 CNC 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(SZGHNCSS) 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 "3-Position 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 "F6-USB-Disk" to open U-disk in program interface, select corresponding program to press "F7-Excut" to execute program, press "Auto" to get into automatic mode and press "Start"/"Run" button to process the program.

Turn "3-Position 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. 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	P15	D15	D14	D13	D12	D11	D10	D9	D8	D7	D6	D5	D4	D3	D2	D1	D0
Axis						Ys	Xs	C			B	A	Z	Y	X		
Default	0	0	0	0	0	0	0	0	0	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

P11: Max Range in B-Negative Direction

P12: Max Range in B-Positive Direction

P12a: Max Range in C-Negative Direction

P12b: Max Range in C-Positive Direction

P12c: Max Range in Xs-Negative Direction

P12d: Max Range in Xs-Positive Direction

P12e: Max Range in Ys-Negative Direction

P12f: Max Range in Ys-Positive Direction

Warning: when shift Metric/Inch, unit is changed(mm/inch), data for limitation won't change, please take care about these.

1.7.1.2 External Switch for limitation

Input Point of Limitation

Mark	Address	Explanation
-L	X09	Limitation in negative direction
+L	X10	Limitation in positive direction

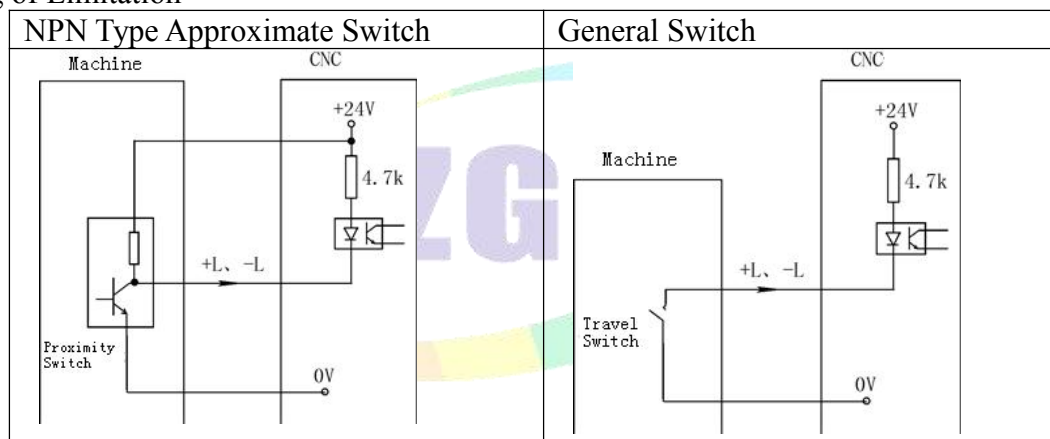
Type of Limitation Switch

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



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 as there are only one input point for all limit switches in negative direction or in positive direction of all axes.

2. CNC system could define home input points as limitation of feeding axis too. 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:

P37, Type of Switch in positive direction, also for +L [0: NO type, 1: NC Type]

P38, Type of Switch in negative direction, also for -L [0: NO type, 1: NC Type]

1.7.1.4 Power ON/OFF CNC

There are two buttons for power on/off CNC controller on C type/E type operation panel. One is Red button, NC type, for power off; another one is Green button, NO type, for power on CNC.

It needs to add one contactor(KM1) for turn on & turn off power of CNC.

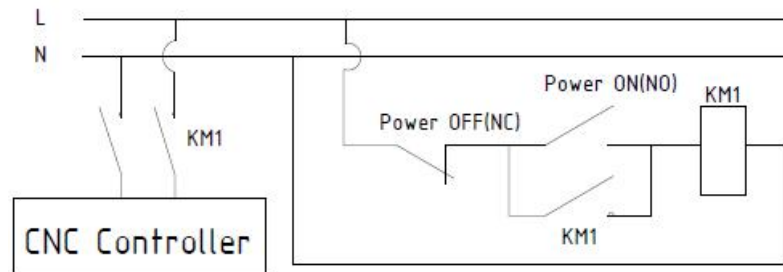


Fig1.7.1 Wiring Diagram for Power ON/OFF

Note: Connect to L&N port of power supply in the rear of CNC controller,220VAC&50/60Hz.



1.8 Diagnosis

Press "Diagnosis" key to enter the diagnosis interface.

Press "F2" & "PgDn/PgUp" or "↵" to check the status of inputs and outputs.

Man Con		N00000								2017-07-04 17:58	
Input point										Program SZGH	
0	0	0	0	0	0	0	0	0	Instruction code		
X00	X01	X02	X03	X04	X05	X06	X07	X08	G53		
T01	T02	T03	T04	T05	T06	T07	T08		T01H0D0		
0	0	0	0	0	0	0	0	0	Machine Status		
X08	X09	X10	X11	X12	X13	X14	X15		M05	M09	M10
M34/A0	-L	+L	M36/Y0	X0	Z0	KRUN	KHALT		M78	M33	M41
0	0	1	0	0	0	0	0	0	G00 X100%		
X16	X17	X18	X19	X20	X21	X22	X23		F100 X100%		
X20	Z20	KLEFT	KRIGHT	STOP	TOK	ALM	ALM1		S0 X 5% SP000		
0	0	0	0	0	0	0	0	0	Machine Coord		
X24	X25	X26	X27	X28	X29	X30	X31		X 0.840 A 0.000		
ALM2	M25	M24	M22	M18	M12	M14	M16		Y 0.000 B 0.000		
1	1	1	1	1	1	1	1	1	Z 0.000		
X32	X33	X34	X35	X36	X37	X38	X39		PartTime 0: 0		
HX	HY	HZ	HA	HX1	HX10	HX100	HOFF		PartNo 0		
0	0	0	0	0	0	0	0	0	SPrpm 0		
X40	X41	X42	X43	X44	X45	X46	X47				
1	1	1	1	0	1	0	1	1			
X60	X61	X62	X63	X64	X65	X66	X67				
DS3	DS2	DS1	DS0	DK3	DK2	DK1	DK0				
No Alarm											
F2 I/O		F3 LAD		F4 ALARM		F5 EdLad		F6 Reset		F8 CANCEL	

Fig1.8.1 System Diagnosis Interface(Input signal-01)

Man Con		N00000								2016-11-28 13:17	
Input point 2										Program 99.TXT	
0	0	0	0	0	0	0	0	0	Instruction code		
X48	X49	X50	X51	X52	X53	X54	X55		G53		
0	0	0	0	0	0	0	0	0	T01H0D0		
0	0	0	0	0	0	0	0	0	Machine Status		
X56	X57	X58	X59	X60	X61	X62	X63		M05	M09	M10
0	0	0	0	0	0	0	0	0	M78	M33	M41
1	1	1	1	0	0	0	0	0	G00 X100%		
X64	X65	X66	X67	X68	X69	X70	X71		F2000 X100%		
VDK0	VDK1	VDK2	VDK3	VDS0	VDS1	VDS2	VDS3		S800 X100% SP000		
0	0	0	0	0	0	0	0	0	Machine Coord		
X72	X73	X74	X75	X76	X77	X78	X79		X 209.696		
KHALT	KRUN	STOP	KLEFT	KRIGHT					Y -47.070		
0	0	0	0	0	0	0	0	0	Z 200.000		
X200	X201	X202	X203	X204	X205	X206	X207		PartTime 0: 0		
HaCOM	HaOFF	HandX	HandY	HandZ	HandA	HandX1	HandX10		PartNo 56		
0	0	0	0	0	0	1	1	1	SPrpm 0		
X208	X209	X210	X211	X212	X213	X214	X215				
HaX100	HaEstop	HandB	HandC	HandXS	HandYS						
No Alarm											
F2 I/O		F3 LAD		F4 ALARM		F5 EdLad		F6 Reset		F8 CANCEL	

Fig1.8.2 System Diagnosis Interface(Input signal-02)

Man Con		N00000		2016-11-28 13:17	
Output Point					
Y00 M61	Y01 M63	Y02 M65	Y03 M67	Y04 M69	Y05 M71
0	0	0	0	0	0
Y08 M32	Y09 M79	Y10 M10	Y11 M08	Y12 M05	Y13 M04
0	0	0	0	0	0
Y16 READY	Y17 INTH	Y18 +T	Y19 -T	Y20 S04	Y21 S03
1	0	0	0	0	0
Y24	Y25	Y26	Y27	Y28 M203	Y29 M204
0	0	0	0	0	0
			Y30 SPEN	Y31 SPEN2	0
Program 99.TXT Instruction code G53 T01H0D0 Machine Status M05 M09 M10 M78 M33 M41 G00 X100% F2000 X100% S800 X100% SP000 Machine Coord X 209.696 Y -47.070 Z 200.000 PartTime 0:0 PartNo 56 SPrpm 0					
No Alarm					
F2 I/O F3 LAD F4 ALARM F5 EdLad F6 Reset F8 CANCEL					

Fig1.8.3 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.

Man Con		N00000		2017-07-04 17:58	
Program SZGH Instruction code G53 T01H0D0 Machine Status M05 M09 M10 M78 M33 M41 G00 X100% F100 X100% S0 X 5% SP000 Machine Coord X 0.840 A 0.000 Y 0.000 B 0.000 Z 0.000 PartTime 0:0 PartNo 0 SPrpm 0					
X28 M295 X28 M123 M122 M225 X25 M282 X26 X27 X28 0Forward M30 S 0000 0Forward M30 R 0001 0003 0004 C-RevL M27 0005 0006 0007 0008					
R---Reload ladder					
No Alarm					
F2 I/O F3 LAD F4 ALARM F5 EdLad F6 Reset F8 CANCEL					

Fig1.8.4 Condition of Inner Register & IOs

Press "PgDn", "PgUp", "Up arrow", "Down Arrow" to check condition of inner registers & outputs & inputs. 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, new PLC after revised &

save will work immediately & no needs to reboot.

"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.

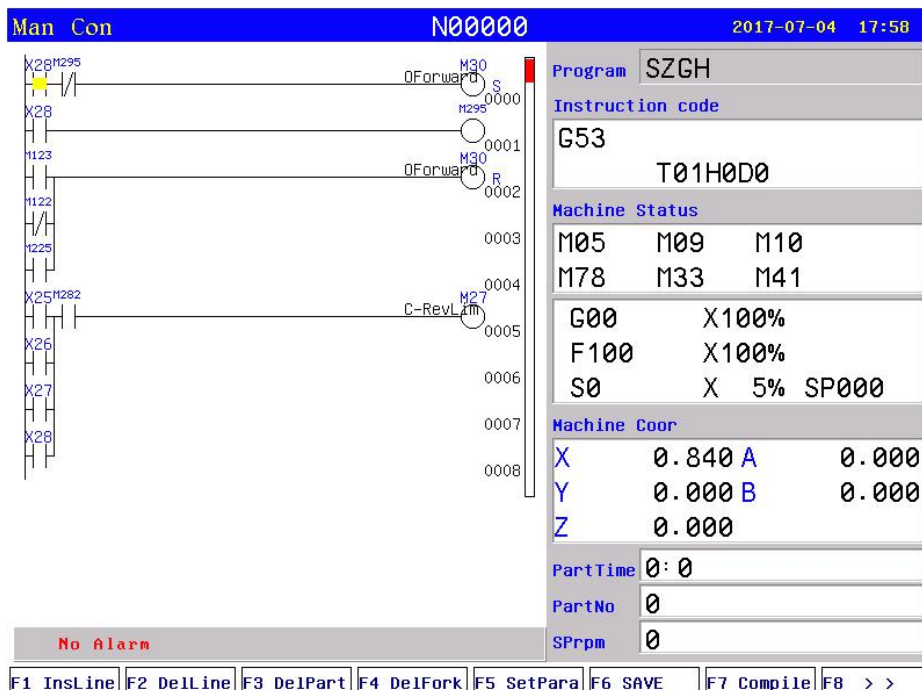


Fig1.8.5 Editing Screen of Inner Ladder

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

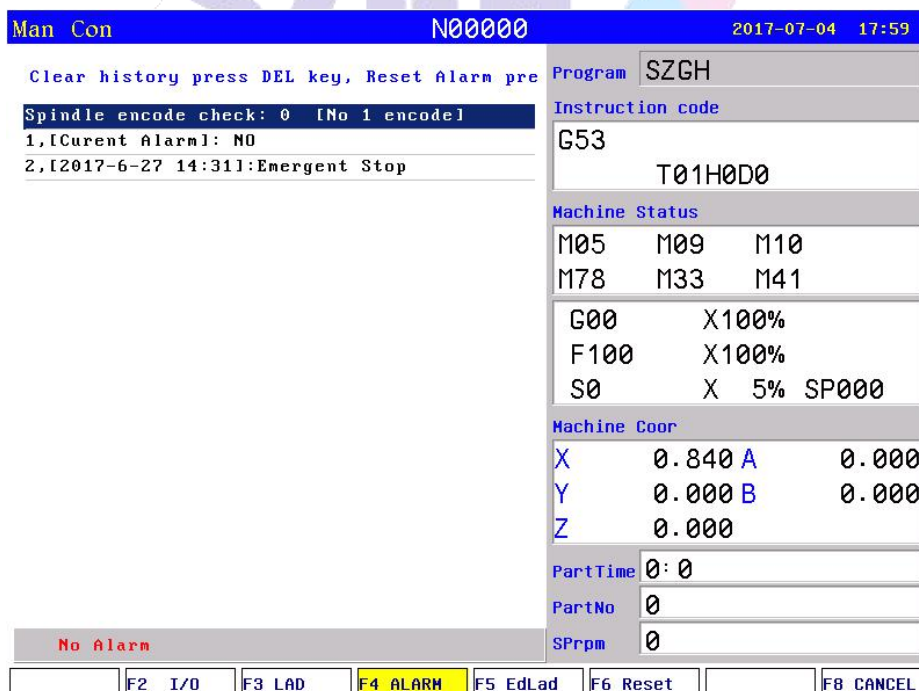


Fig1.8.6 System Diagnosis Interface(Alarm messages)

Spindle encode check:0 ,shows current resolution of No.1 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]

CNC will only record recently 10pcs of alarm messages.

1.9 Programming Operation

Press "Program" to enter into status of Program as following.

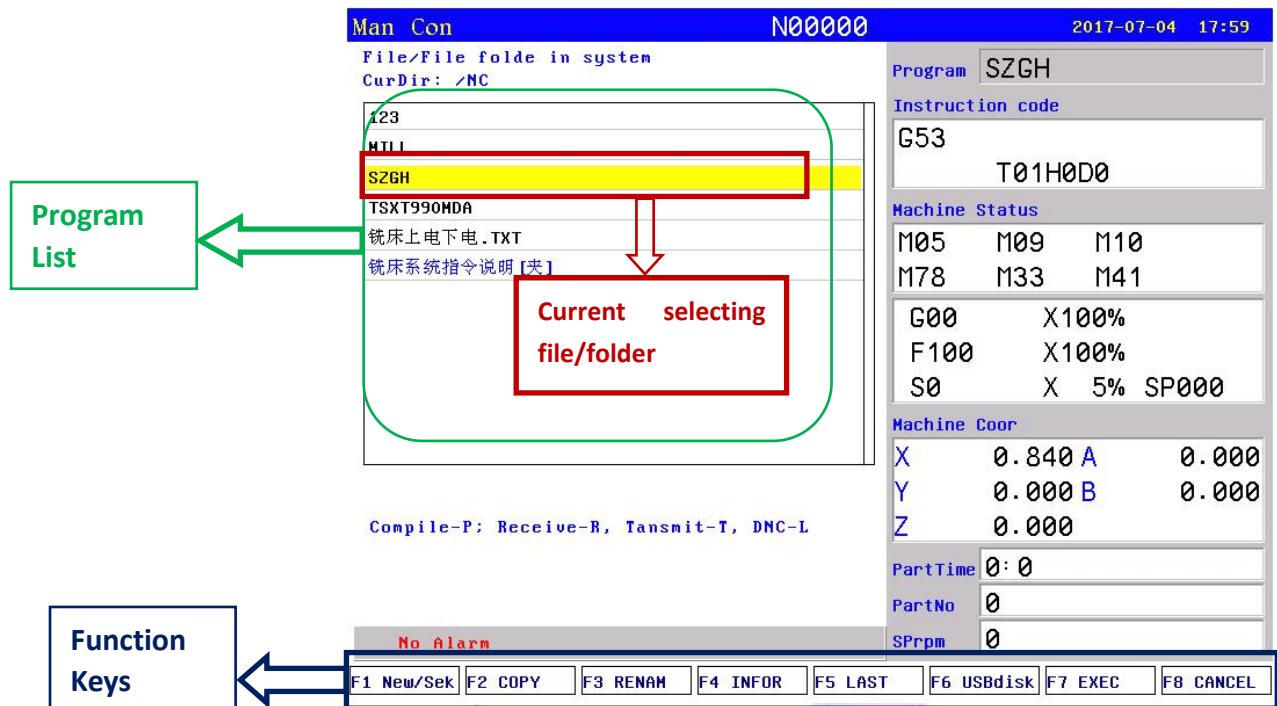


Fig1.9.1 Interface of Program

Management of program adopt mode of file/folder management, storage room of SZGH CNC system is 128Mb, which can save 32Mb program, and 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.

Man Con		N00000		2017-07-04 17:59	
Edit: /NC/SZGH			Program SZGH		
N1 G00 X0 Z0 N2 G00 X200 Z200 N3 G00 U-44 N4 W-5 N5 G01 U4 W-2 F300 N6 W-11 N7 U-4 W-2 N8 W-5 N9 U10 W-10 N10 W-6 N11 G02 U-6 W-9 I12 K-9 F5 N12 U10 W-15 I25 K0 F300			Instruction code G53 T01H0D0		
			Machine Status		
			M05 M09 M10 M78 M33 M41		
			G00 X100% F100 X100% S0 X 5% SP000		
			Machine Coord		
			X 0.840 A 0.000 Y 0.000 B 0.000 Z 0.000		
			PartTime 0:0		
			PartNo 0		
			SPrpm 0		
No Alarm					
F1 COMPIL		F2 FrELine		F3 TeachIn	
F4 POS		F6 DeLine		F7 >>	
				F8 CANCEL	

Fig1.9.2 (1)Editing interface of Program

Man Con		N00000		2017-07-04 17:59	
Edit: /NC/SZGH			Program SZGH		
N1 G00 X0 Z0 N2 G00 X200 Z200 N3 G00 U-44 N4 W-5 N5 G01 U4 W-2 F300 N6 W-11 N7 U-4 W-2 N8 W-5 N9 U10 W-10 N10 W-6 N11 G02 U-6 W-9 I12 K-9 F5 N12 U10 W-15 I25 K0 F300			Instruction code G53 T01H0D0		
			Machine Status		
			M05 M09 M10 M78 M33 M41		
			G00 X100% F100 X100% S0 X 5% SP000		
			Machine Coord		
			X 0.840 A 0.000 Y 0.000 B 0.000 Z 0.000		
			PartTime 0:0		
			PartNo 0		
			SPrpm 0		
No Alarm					
F1 DeIBlock		F2 CopBlock		F3 ARRAY	
F4 SEARCH		F5 ALTER		F6 AALTER	
				F7 <<	
				F8 CANCEL	

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" key 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

"F4"key 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" key, 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 "F5-Last"key 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 "F7-Exec"key to execute 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.

The screenshot displays the 'Main Con' interface with 'N00000' and a timestamp of '2017-07-04 17:59'. The left pane shows a file list under 'File/File folde in USB disk' with 'CurDir: /USB'. The file '000ASITHA [关]' is selected. The right pane shows 'Program SZGH' with 'Instruction code G53 T01H0D0'. Below this, 'Machine Status' shows M05, M09, M10, M78, M33, M41, and G00, F100, S0 with their respective values. 'Machine Coord' shows X, Y, Z coordinates. 'PartTime', 'PartNo', and 'SPrpm' are also displayed. A 'No Alarm' status is shown at the bottom. The bottom menu bar includes: F1 Backup, F2 Restore, F3 Export, F4 Import, F5 LAST, F6 SYSTM, F7 EXEC, F8 CANCEL.

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 current 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 data 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.

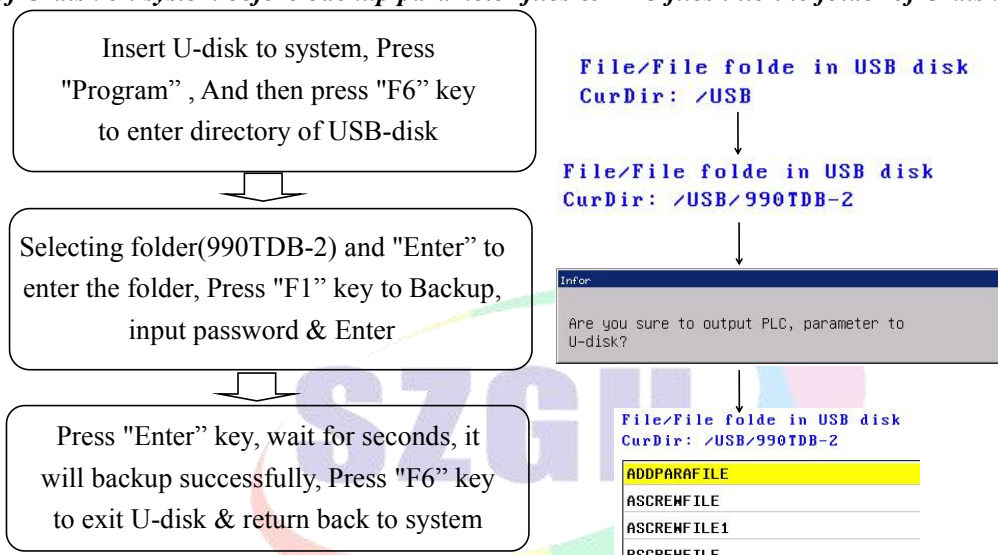


Fig1.9.5 Steps of Backup PLC & Parameter to U-disk

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.

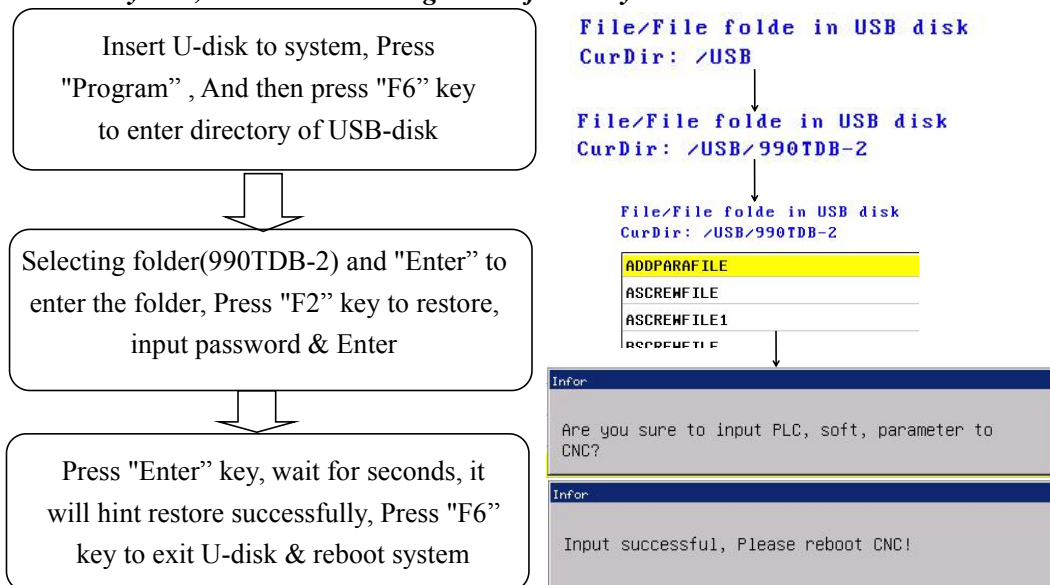


Fig1.9.6 Steps of Restore PLC & Parameter to U-disk

Chapter 2 Parameter List

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

Man Con		N00000		2017-07-04 18:00			
1,Cycle d of G7		5.000		Program SZGH			
2,Cycle d of G8		5.000		Instruction code			
3,Cycle Q-direc		1		G53			
4,Cycle Q-direc		1		T01H0D0			
5,Cycle spindle		0		Machine Status			
6,Mode of G84G7		0.000		M05 M09 M10			
7,Cycle d of G8		0.000		M78 M33 M41			
17,Running prog		0		G00 X100%			
18,Set M20 the		-1		F100 X100%			
19,Set part cou		0		S0 X 5% SP000			
21,G01/G02/G03		0		Machine Coor			
22,G00 line del		0		X 0.840 A 0.000			
23,Smooth accel		70		Y 0.000 B 0.000			
				Z 0.000			
				PartTime 0: 0			
				PartNo 0			
				SPrpm 0			
No Alarm							
F1 User	F2 Speed	F3 Axis	F4 Tool	F5 Other	F6 Coor	F7 PASSWD	F8 CANCEL

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 screen & select kind of parameter list, press "Down arrow", "Up arrow", "PgDn"(Page down), "PgUp"(page up) 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 V10.18.

3.These parameter list are for 8 axes CNC system in the manual.And there are only parameters for related axes CNC system. Eg.:If 4 Axes CNC,only parameters for 4 axes(XYZA).

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)	0.00
7	(d)Escaping Amount of G74/G84 (mm)	0.00
8	Pause Time Before Raise Up Tool at G73G83(ms)[Abs>5,-:G01;+:G00]	0
17	Interlock between Running Program & SP-Rotating [1:Yes, 0:No]	0
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
202	System Inner Parameter(cannot alter)	0
203	Inspect over-cut when tool radius offset (2131970Yes,34818:No, 6326274:Pre-read)	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 (D2-D15: X16-X31)	1
231	Mode of "Delete" key [0:backward deletion,1:Forward delete]	0
232	Detect SP_Zero position before tapping [18:Yes,0:No]	0
233	G06 Circle teaching function[0:No, 1:Yes]	1
234	Activate Program Back Function with Handwheel[+8:Yes, Other: No]	1
235	Automatically generate comments when teaching [0:Yes,1:No]	0
236	Type of Arc code when Teach-in[0:G5/G6 IJK,1:G2/G3 R]	0
237	Mode of Generate G code in Teach_In[0:Normal,1:G01]	0
238	Auto Insert one line in middle line during Teach_In[0:Yes,1:Alter]	0
307	M18xx/M28xx/WAT alarm time(ms)[>=10]	1
400	Translate DXF file to G code[1:Sequencing,4:Start point ,8:No sequencing]	0
401	DXF_X-axis coordinate of Start Point	0
402	DXF_Y-axis coordinate of Start Point	0
500	G74 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]	0
501	G81 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]	0
502	G82 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]	0
503	G83 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]	0
504	G84 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]	0
551	Machine structure of 5 Axes CNC machine tool(10-99)	1
552	Reverse direction of compensation with RTCP function(BX+4;BZ+8;AY+16;AZ+32)	1

553	Y-Axis machine coordinate value on A-axis rotary center(um)	1
554	Z-Axis machine coordinate value on A-axis rotary center(um)	1
555	Length of tool when measure rotary center of A-axis(um)	1
556	X-Axis machine coordinate value on C-axis rotary center(P551=12)	0
557	AC Axis offset of tool return Transformation of AC Axis (um)	0
600	Internal multiplier of non-G00 instruction[>=20 valid]	0

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 do 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-elation. the smaller it is,the faster the acc/dec-elation 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 [≥ 2 minutes]

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,Inspect overcut when G41/G42 tool radius offset (2131970:Yes,34818:No, 6326274:Pre-read)

It is set for if inspect overcut when use G41/G42 tool radius compensation, when set to 2131970, which means strict inspect; when set to 2131970,No inspect, when set to 6326274, CNC is on fast pre-read mode processing.

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 (D2-D15: X16-X31)

CNC system support run processing program by input points , related input points is X16-X23 & X26-X31.D2-D7:X26-X31; D8-D15:X16-X23.

D2 bit=1,also +4 , means X26 input is valid, execute program of "X26"/"HIDEFILEX26"

D3 bit=1,also +8, means X27 input is valid, execute program of "X27"/"HIDEFILEX27"

D4 bit=1,also +16, means X28 input is valid, execute program of "X28"/"HIDEFILEX28"

D5 bit=1,also +32, means X29 input is valid, execute program of "X29"/"HIDEFILEX29"

D6 bit=1,also +64, means X30 input is valid, execute program of "X30"/"HIDEFILEX30"

D7 bit=1,also +128, means X31 input is valid, execute program of "X31"/"HIDEFILEX31"

D8 bit=1,also +256 , means X16 input is valid, run program of "X16"/"HIDEFILEX16"

D9 bit=1,also +512, means X17 input is valid, run program of "X17"/"HIDEFILEX17"

D10 bit=1,also +1024 , means X18 input is valid, run program of "X18"/"HIDEFILEX18"

D11 bit=1,also +2048,means X19 input is valid, run program of "X19"/"HIDEFILEX19"

D12 bit=1,also +4096,means X20 input is valid, run program of "X20"/"HIDEFILEX20"

D13 bit=1,also +8192,means X21 input is valid, run program of "X21"/"HIDEFILEX21"

D14 bit=1,also +16384,means X22 input is valid, run program of "X22"/"HIDEFILEX22"

D15 bit=1,also +32768, means X23 input is valid, run program of "X23"/"HIDEFILEX23"

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.

235,Automatically generate comments when teaching [0:Yes,1:No]

It sets for if generate comments automatically when user adopt teach-in function, 0: Yes, CNC will generate comments automatically when record of Teach-in.

236,Type of Arc code when Teach-in[0:G5/G6 IJK,1:G2/G3 R]

It sets type of G code format for arc when user adopt Teach-in function, 0: when arc during teach-in, use G5/G6 & IJK format; 1: G2/G3 & R format.

237,Mode of Generate G code in Teach_In[0:Normal,1:G01]

It sets mode of generate G code in Teach-in function, 0:Normal, 1: Generate G01 in each one linear movement.

238,Auto Insert one line in middle line during Teach_In[0:Yes,1:Alter]

It sets if insert one line automatically when do teach-in during middle line,0:Yes, insert one line automatically, 1:alter current middle line.

307,M18xx/M28xx/WAT alarm time(ms)[>=10]

It sets max waiting time for M18xx/M28xx, when over max time,CNC will hints alarm.unit is ms. Example set to 100 (>=10), also 1000ms, when M1810 code,detect X10 is on within 1000ms, otherwise CNC hints alarm.

400,Translate DXF file to G code[1:Sequencing,4:Start point ,8:No sequencing]

401,DXF_X-axis coordinate of Start Point

402,DXF_Y-axis coordinate of Start Point

1)When P400=1, CNC will make sequencing according to start/end point of dxf file

2)When P400=4, After make sequencing according to according to start/end point of dxf file, and compare distance with starting point(P401&P402),starting from nearest point.

3) When P400=8, CNC doesn't make sequencing automatically. When use AutoCAD to make drawing & generate dxf file, we need to take care of sequence of drawing, as dxf files are saved with drawing sequence.So CNC generate G code files also according to this sequence.

500,G74 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

501,G81 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

502,G82 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

503,G83 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

504,G84 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

These parameters are for user define corresponding G codes for G74-G84.Which is for select Non-mode/mode status,and user-defined.

551,Machine structure of 5 Axes CNC machine tool(10-99)

552,Reverse direction of compensation with RTCP function(BX+4;BZ+8;AY+16;AZ+32)

553,Y-Axis machine coordinate value on A-axis rotary center(um)

554,Z-Axis machine coordinate value on A-axis rotary center(um)

555,Length of tool when measure rotary center of A-axis(um)

556,X-Axis machine coordinate value on C-axis rotary center(P551=12)

557,AC Axis offset of tool return Transformation of AC Axis (um)

These parameters for for RTCP function with different structure of 5 Axes CNC machine. Please check Chapter 4.10 on Programming Book.

Note: It is only works on CNC system with 5 axes at least.

600,Internal multiplier of non-G00 instruction[>=20 valid]

It is set inner rate of G codes except G00.It is valid only when its value>=20.

Note:XYZABCXSYS display at parameter list, X:1st axis, Y:2nd axis,Z:3rd axis, A:4th axis, B:5th axis, C:6th axis, XS:7th axis,YS:8th axis.We write short name with XYZABCXSYS for feeding axes.Example: even if we can alter B-axis display to C-axis,we still use B-axis stands for 5th axis.use C-axis stands for 6th axis.



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	G00 Speed of B-axis (mm/min)	4000.000
5a	G00 Speed of C-axis (mm/min)	4000.000
5b	G00 Speed of Xs-axis (mm/min)	4000.000
5c	G00 Speed of Ys-axis (mm/min)	4000.000
6	Manual Max Feeding Speed(mm/min)	10000.000
6-1	X_Manual Max Feeding Speed(mm/min)[>=50]	0
6-2	Y_Manual Max Feeding Speed(mm/min)[>=50]	0
6-3	Z_Manual Max Feeding Speed(mm/min)[>=50]	0
6-4	A_Manual Max Feeding Speed(mm/min)[>=50]	0
6-5	B_Manual Max Feeding Speed(mm/min)[>=50]	0
7	Auto Max Feeding Speed(mm/min)	15000.000
8	Default Speed of G01/G02/G03 (mm/min)	2000.000
9	Running Speed at Simulation Mode (mm/min)	10000.000
10	Manual Feeding Speed(mm/min)	1000.00
11	Manual Spindle Speed (rpm)	0.000
12	Take-off speed(mm/min)	500
13	Maximum mutation speed(mm/min)	500
14	Limit G01/G02/G03 Speed of each axis [1:Yes, 0:No]	0
15	Max Speed of X_G1/G2/G3 (mm/min)	2000
16	Max Speed of Y_G1/G2/G3 (mm/min)	2000
17	Max Speed of Z_G1/G2/G3 (mm/min)	2000
18	Max Speed of A_G1/G2/G3 (mm/min)	2000
19	Max Speed of B_G1/G2/G3 (mm/min)	2000
19a	Max Speed of C_G1/G2/G3 (mm/min)	2000
19b	Max Speed of Xs_G1/G2/G3 (mm/min)	2000
19c	Max Speed of Ys_G1/G2/G3 (mm/min)	2000
20	X_Acceleration/Deceleration Constant [1~99999]	50000
21	Y_Acceleration/Deceleration Constant [1~99999]	50000
22	Z_Acceleration/Deceleration Constant [1~99999]	50000
23	A_Acceleration/Deceleration Constant [1~99999]	50000
24	B_Acceleration/Deceleration Constant [1~99999]	50000
24a	C_Acceleration/Deceleration Constant [1~99999]	50000
24b	Xs_Acceleration/Deceleration Constant [1~99999]	50000
24c	Ys_Acceleration/Deceleration Constant [1~99999]	50000
25	Acceleration/Deceleration Constant When Auto Run	2
26	Handwheel_Acce/Dece-leration Constant [500--32000]	6000
27	Handwheel_Acce/Dece-leration Constant when dry [>500]	60000
28	G00 Speed when dry (mm/min) [>10]	1500
28-1	Run program Handwheel G00 start speed(mm/min)[>5]	100
29	X_Max Speed with Handwheel (mm/min)	2000
30	Y_Max Speed with Handwheel (mm/min)	2000
31	Z_Max Speed with Handwheel (mm/min)	2000
32	A_Max Speed with Handwheel (mm/min)	2000
33	B_Max Speed with Handwheel (mm/min)	2000
33a	C_Max Speed with Handwheel (mm/min)	2000
33b	Xs_Max Speed with Handwheel (mm/min)	2000

33c	Ys Max Speed with Handwheel (mm/min)	2000
33-1	Make thread Z acceleration(Servo,=0)	0.000
33-2	Make thread X acceleration(Servo,=0)	50000
33-3	X-Retract Ratio with servo threading Cycle(*0.1)	4000
34	Type of Acceleration/Deceleration [0 mean line,8 mean curve]	0
35	Initial Acce/Dece-leration Constand when curve [≥ 10]	100
36	Quadratic Acce/Dece-leration Constant when curve [≥ 10]	500
37	Max Acce/Dece-leration Constant when curve [≥ 500]	1
38	X Homing Speed (mm/min)	3000.000
39	X Speed for off switch (mm/min)	250.000
40	Y Homing Speed (mm/min)	3000.000
41	Y Speed for off switch (mm/min)	250.000
42	Z Homing Speed (mm/min)	3000.000
43	Z Speed for off switch (mm/min)	250.000
44	A Homing Speed (mm/min)	3000.000
45	A Speed for off switch(mm/min)	250.000
46	B Homing Speed (mm/min)	2500.000
47	B Speed for off switch (mm/min)	250.000
47-1	C Homing Speed (mm/min)	3000.000
47-2	C Speed for off switch (mm/min)	250.000
47-3	Xs Homing Speed (mm/min)	3000.000
47-4	Xs Speed for off switch (mm/min)	250.000
47-5	Ys Homing Speed (mm/min)	2500.000
47-6	Ys Speed for off switch (mm/min)	250.000
48	Max Speed of 1st Spindle (rpm)	3000
49	Max Speed of Spindle at 2nd gear (rpm)	3000
50	Max Speed of Spindle at 3rd gear (rpm)	3000
51	Max Speed of Spindle at 4th gear (rpm)	3000
52	Max Speed of 2nd Spindle (rpm)	3000
52-1	Max Speed of 3rd Spindle (rpm)	1
52-2	Max Speed of 4th Spindle (rpm)	1
52-3	Manual Speed of 3rd Spindle(rpm)	0
52-4	Manual Speed of 4th Spindle (rpm)	0
53	Compensation Mode of Arc Backlash (0: A, 8: B)	8
54	Compensation Speed in B Mode (mm/min)	3000
54-1	Beginning Compensation Speed in B Mode (mm/min)	2000
54-2	Acce/Dece-leration Constant in B mode (mm/min/s)	60000
55	Activate Speed Processing Function[1:Yes, 0:No]	1
55-1	End speed when reverse&decrease speed during running(mm/min)	200
56	Beginning Speed when hardware limitation(mm/min)	1.000
57	Handwheel Stopping Speed (mm/min)[>100]	100
58	SP_Speed when exit at following tapping mode(rpm)[>1]	1
58-1	SP Reverse Backlash Compensation when tapping(Pulse)	1
58-2	Advance Retired Value before reverse rotation of following tapping(um)[10-5000]	1
58-3	Retired Speed when tapping (mm/min)[≥ 60]	1
60	Activate Smooth Processing Function(+4:Maual,+8:MPG,+16:Program)	28
61	Time constant of smooth running on Manual[2-500]	50
62	Time constant of smooth running on MPG[2-500]	50
63	Time constant of smooth running on Program[2-500]	50
65	Manual enhancement smoothing processing time constant [2-50]	0
66	Handwheel enhancement smoothing processing time constant [2-50]	0
67	Program enhancement smoothing processing time constant [2-50]	0
68	Grade of smoothing process [1-9](Bigger, higher smooth)	0
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
106	C_Beginning Speed (mm/min)[>1]	500
107	Xs_Beginning Speed (mm/min)[>1]	500
108	Ys_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
116	C_Max Speed Increment (mm/min)[>1]	500
117	Xs_Max Speed Increment (mm/min)[>1]	500
118	Ys_Max Speed Increment (mm/min)[>1]	500
200	Coherent movement is valid for G00[1:No,16:Yes]	1
210	Wait SP_Speed smooth when threading	0
211	SP_Starting Speed Tapping-Feed(rpm)	1
212	SP_Starting Speed Tapping-Extract(rpm)	1
213	Acceleration-Feed-Rigid Tapping(rpm/S)[>1]	1
214	Acceleration-Retreat-Rigid Tapping(rpm/S)[>1]	1
215	Reserve-Feed-Rigid Tapping(1/1000Rev)[>2]	1
216	Trailing-Feed-Rigid Tapping(1/1000Rev)[>2]	1
217	Reserve-Retreat-Rigid Tapping(1/1000rev)[>2]	1
218	Trailing-Retreat-Rigid Tapping(1/1000rev)[>2]	1
219	Time Constant of Smooth processing for rigid tapping[20002-20500]	1
230	G00 Speed of SP-axis (mm/min)	1000
231	Mode for SP-axis[+4:F,+8:G90/G91,+16:Pulse]	10
232	Orientation Direction of SP-axis[0:Pos,1:Neg,2:near]	2
233	Control Mode of SP_homing	2
234	Offset Degrees when SP_Orientation(0.1Deg)[Neg:Non-Oriente]	0
235	SP_Homing Speed(0.1 rpm)	800

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)
 5,G00 Speed of B-axis (mm/min) 5a,G00 Speed of C-axis (mm/min)
 5b,G00 Speed of Xs-axis (mm/min) 5c,G00 Speed of Ys-axis (mm/min)

It is speed of G00, also rapid speed(when press Rapid key on panel) of feeding axis, Max is 30000 (unit:mm/min).

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

6,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.

7,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.

8,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)

9,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)

10,Manual Feeding Speed(mm/min)

It is the speed of feeding axis in Manual.Range:< max feeding speed (also P6)

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

11,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.

12 Take-off speed(mm/min)

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.

13 Maximum mutation speed(mm/min)

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.

14,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.

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

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

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

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

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

19a,Max Speed of Y_G1/G2/G3 (mm/min)

19b,Max Speed of X_G1/G2/G3 (mm/min)

19c,Max Speed of Z_G1/G2/G3 (mm/min)

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

20,X_Acceleration/Deceleration Constant

21,Y_Acceleration/Deceleration Constant

22,Z_Acceleration/Deceleration Constant

23,A_Acceleration/Deceleration Constant

24,B_Acceleration/Deceleration Constant

24a,C_Acceleration/Deceleration Constant

24b,Xs_Acceleration/Deceleration Constant

24c,Ys_Acceleration/Deceleration Constant

It is time constant of feeding xis acc/dec-elation, the bigger it is ,the faster the ace/dec-elation is. Unit: (mm/min)/s, range is 1~99999

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

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

It is for set constant of acc/deceleration 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.

26, Handwheel_Acc/Deceleration Constant [500--32000]

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

27, Handwheel_Acc/Deceleration Constant when dry [>500]

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

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

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

29, X_Max Speed with Handwheel (mm/min) 30, Y_Max Speed with Handwheel (mm/min)

31, Z_Max Speed with Handwheel (mm/min) 32, A_Max Speed with Handwheel (mm/min)

33, B_Max Speed with Handwheel (mm/min) 33a, C_Max Speed with Handwheel (mm/min)

33b, Xs_Max Speed with Handwheel (mm/min) 33c, Ys_Max Speed with Handwheel (mm/min)

It is for limit max speed of feeding axis when use handwheel.

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

33-1, Make thread Z acceleration (Servo, =0)

It sets acceleration & deceleration constant of Z-axis when threading. When use servo, set to 0.

33-2, Make thread X acceleration (Servo, =0)

It sets acceleration & deceleration constant of X-axis when threading. When use servo, set to 0.

33-3, X-Retract Ratio with servo threading Cycle (*0.1)

It set retract ratio of X-axis when do threading cycle with servo system.

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

It sets type of acc/deceleration. 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.

35, Initial Acc/Deceleration Constant when curve [≥ 10]

It is initial acc/deceleration constant when P27 set curve type. Range: ≥ 10 .

36, Quadratic Acc/Deceleration Constant when curve [≥ 10]

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

37, Max Acc/Deceleration Constant when curve [≥ 500]

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

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

38,X_Homing Speed (mm/min)	39,X_Speed for off switch (mm/min)
40,Y_Homing Speed (mm/min)	41,Y_Speed for off switch (mm/min)
42,Z_Homing Speed (mm/min)	43,Z_Speed for off switch (mm/min)
44,A_Homing Speed (mm/min)	45,A_Speed for off switch (mm/min)
46,B_Homing Speed (mm/min)	47,B_Speed for off switch (mm/min)
47-1,Y_Homing Speed (mm/min)	47-2,Y_Speed for off switch (mm/min)
47-3,Z_Homing Speed (mm/min)	47-4,Z_Speed for off switch (mm/min)
47-5,A_Homing Speed (mm/min)	47-6,A_Speed for off switch (mm/min)

P38/P40/P42/P44/P46/P47-1/P47-3/P47-5 are set for homing speed of feeding axis. Unit: mm/min. the range is less than G00 speed.

P39/P41/P43/P45/P47-2/P47-4/P47-6 are set for speed for off switch after feeding axis reach at homing switch. & Speed for detecting Z0 phase encoder signal. 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.

48, 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).

49, 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.

50, 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.

51, 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.

52, 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.

52-1, Max Speed of 3rd Spindle (rpm)

It is the max speed of 3rd spindle, it is also the speed of Xs axis output pulse+direction signal. Specified by "SC#", unit is rpm.

52-2, Max Speed of 4th Spindle (rpm)

It is the max speed of 4th spindle, it is also the speed of Ys axis output pulse+direction signal. Specified by "SD#", unit is rpm.

52-3,Manual Speed of 3rd Spindle(rpm)

It is the manual speed of 3rd spindle,it is also the speed of Xs axis output pulse+direction signal. Unit is rpm.

52-4,Manual Speed of 4th Spindle (rpm)

It is the manual speed of 4th spindle,it is also the speed of Ys axis output pulse+direction signal. Unit is rpm.

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

It is compensation mode for arc reverse backlash.

0(D3=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 10000mm/min.

+8(D3=1): means B compensation mode , which is that the compensation speed is set by P54 in the following.

+1(D0=1):Length offset compensation is B mode,otherwise A mode. If A mode, length offset always make compensation at Z-axis. If B mode, length offset compensation are based on G17/G18/G19. G17: compensate Z-axis ; G18: compensate Y-axis; G19:compensate Z-axis.

+2(D1=1): mode of canned cycle is B mode,otherwise A mode. If A mode, CNC always make compensation at Z-axis. If B mode, compensation are based on G17/G18/G19. G17: compensate Z-axis ; G18: compensate Y-axis; G19:compensate Z-axis.

+4(D2=1): 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.

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

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

54-1,Beginning Compensation Speed in B mode (mm/min)

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

54-2,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.

55,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.

55-1,End speed when reverse&decrease speed during running(mm/min)

It is set ending speed when CNC is running with reverse direction & decreasing speed, unit:mm/min.

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

It sets beginning speed for deceleration when feeding axes touch hardware limit.When CNC system is configured with servo, no need to set beginning speed , set to 1 normally.

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

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

58,SP_Speed when exit at tapping with following mode

It is min speed before spindle reverse rotation when tapping.

58-1,SP_Reverse Backlash Compensation when tapping

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

58-2,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

58-3,Retired Speed when tapping (mm/min)[>=60]

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

60,Activate Smooth Processing Function(+4:Manual,+8:MPG,+16:Program)

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] 106,C_Beginning Speed (mm/min)[>1]

107,Xs_Beginning Speed (mm/min)[>1] 108,Ys_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] 116,C_Max Speed Increment (mm/min)[>1]

117,Xs_Max Speed Increment (mm/min)[>1] 118,Ys_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.

211,SP_Starting Speed_Tapping-Feed(rpm)

It is starting speed of spindle axis for feeding when tapping.

212,SP_Starting Speed_Tapping-Extract(rpm)

It is starting speed of spindle axis for extract when tapping

213,Acceleration-Feed-Rigid Tapping(rpm/S)[>1]

It is acceleration/deceleration for spindle feeding when rigid tapping,unit:rpm/s.

214,Acceleration-Retreat-Rigid Tapping(rpm/S)[>1]

It is acceleration/deceleration for spindle retreat when rigid tapping,unit:rpm/s.

215,Reserve-Feed-Rigid Tapping(1/1000Rev)[>2]

It is reserve for spindle feed during rigid tapping. Unit:1/1000r, Range:>2

216,Trailing-Feed-Rigid Tapping(1/1000Rev)[>2]

It is trailing for spindle feed during rigid tapping. Unit:1/1000r, Range:>2

217,Reserve-Retreat-Rigid Tapping(1/1000rev)[>2]

It is reserve for spindle retreat during rigid tapping. Unit:1/1000r, Range:>2

218,Trailing-Retreat-Rigid Tapping(1/1000rev)[>2]

It is trailing for spindle retreat during rigid tapping. Unit:1/1000r, Range:>2

219,Time Constant of Smooth processing for Rigid tapping[20002-20500]

It is time constant of smooth processing for rigid tapping, range of value is 20002-20500.

230,G00 Speed of SP-axis (0.1rpm)

It is G00 speed for SP-axis,which is on CN10 plug,unit:0.1rpm.

231,Mode for SP-axis[+4:F,+8:G90/G91,+16:Pulse]

It set mode for SP-axis & other special function. +4: run with F code, +8: run with G90/G91

code, +16: send pulse with display value, +8192:CNC output analog voltage per one second, +16384: run M05 & turn off analog voltage output.

232,Orientation Direction of SP-axis[0:Pos,1:Neg,2:Near]

It set orientation direction of sp-axis when rigid tapping with interplote mode.0: Positive direction, 1: Negative direction, 2: nearest direction

233,Homing Control Mode of SP-Axis

It set homing control mode of SP-axis, 1: Controlled by Pulse(CN10 plug+spindle encoder), 2: Control by Spindle driver(output M61, detect M22 is valid), 16 or 32: after orientation, move SP to degree of P234, 16: Output M75: detect M22 is valid.

234,Offset Degrees when SP_Orientation(0.1Deg)[Neg:Non-Orientation]

It set offset degrees after orientation end, unit:0.1degree, if negative number, don't orientation.

235,SP_Homing Speed(0.1 rpm)

It set homing speed of Sp-axis,unit: 0.1rpm.



2.3 Axis parameter

P	Axis Parameter	Ex-Value
1	Switch Type for Feed-Rate [0: Key, 1: Band Switch]	1
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	Max Travel in B_Negative direction (mm)	-9999.000
12	Max Travel in B_Positive direction (mm)	9999.000
12a	Max Travel in C_Negative direction (mm)	-9999.000
12b	Max Travel in C_Positive direction (mm)	9999.000
12c	Max Travel in Xs_Negative direction (mm)	-9999.000
12d	Max Travel in Xs_Positive direction (mm)	9999.000
12e	Max Travel in Ys_Negative direction (mm)	-9999.000
12f	Max Travel in Ys_Positive direction (mm)	9999.000
13	SP_Braking Time (10ms)	150
14	SP_Braking is Long Signal [0: No, 1: Yes]	0
15	Soft-Limitation is valid [0: Yes, 1: No][D2X;D3C(Y);D4Z;D5A;D6B]	0101000100000001
16	X_Reverse Backlash Compensation (um)	0
17	Y_Reverse Backlash Compensation (um)	0
18	Z_Reverse Backlash Compensation (um)	0
19	A_Reverse Backlash Compensation (um)	0
20	B_Reverse Backlash Compensation (um)	0
20a	C_Reverse Backlash Compensation (um)	0
20b	Xs_Reverse Backlash Compensation (um)	0
20c	Ys_Reverse Backlash Compensation (um)	0
21	X_Direction [1:normal, 0: Reverse]	0
22	Y_Direction [1:normal, 0: Reverse]	0
23	Z_Direction [1:normal, 0: Reverse]	0
24	A_Direction [1:normal, 0: Reverse]	0
25	B_Direction [1:normal, 0: Reverse]	0
25a	Direction for C/Xs/Ys axis(+2:C, +4:Xs, +8:Ys)	0
26	Using Electron Gear Ratio for Feeding Axes [0: Yes, 1: No]	0
27	Numerator of X_Electron Gear	1
28	Denominator of X_Electron Gear	1
29	Numerator of Y_Electron Gear	1
30	Denominator of Y_Electron Gear	1
31	Numerator of Z_Electron Gear	1
32	Denominator of Z_Electron Gear	1
33	Numerator of A_Electron Gear	1
34	Denominator of A_Electron Gear	1
35	Numerator of B_Electron Gear	1
36	Denominator of B_Electron Gear	1
36a	Numerator of C_Electron Gear	1
36b	Denominator of C_Electron Gear	1

36c	Numerator of Xs_Electron Gear	1
36d	Denominator of Xs_Electron Gear	1
36e	Numerator of Ys_Electron Gear	1
36f	Denominator of Ys_Electron Gear	1
37	Type of Limit Switch in Positive Direction[0:NO type, 1:NC type]	0
38	Type of Limit Switch in Negative Direction[0:NO type, 1:NC type]	0
39	Type of Home [D3X;D4(C)Y;D5Z;D6A;D7B;D8C;D9XS;D10YS; 0: Switch; 1:float Zero, D0D1:Y-axis Coordinate]	0000011111111011
40	X_Machine Coordinate of float zero point	
41	Y_Machine Coordinate of float zero point	
42	Z_Machine Coordinate of float zero point	
43	A_Machine Coordinate of float zero point	
44	B_Machine Coordinate of float zero point	
44a	C_Machine Coordinate of float zero point	
44b	Xs_Machine Coordinate of float zero point	
44c	Ys_Machine Coordinate of float zero point	
45	Grade of Homing[1:No,0:Prompt,8:Compulsion,9:Super compulsion]	9
46	Mode of Homing	1
47	Direction of Homing[D2:X,D3:C(Y),D4:Z,D5:A,D6:B]	0000000000000001
48	Type of Switches for Homing	00000000
49	Range of Detect Z0 in X axis (100um)	100
50	Range of Detect Z0 in Y axis (100um)	100
51	Range of Detect Z0 in Z axis (100um)	100
52	Range of Detect Z0 in A axis (100um)	100
53	Range of Detect Z0 in B axis (100um)	100
54	Offset after homing in X axis	0
55	Offset after homing in Y axis	0
56	Offset after homing in Z axis	0
57	Offset after homing in A axis	0
58	Offset after homing in B axis	0
58-1	Offset after homing in C axis	0
58-2	Offset after homing in Xs axis	0
58-3	Offset after homing in Ys axis	0
59	Spindle is rotating when shift gear[1:Yes, 0:No]	0
60	Rotating Speed of Spindle when shift gear(1/100rpm)	1000
61	Rotating Direction of Spindle when shift gear[0:CW, 1:CCW]	1
62	Pause Time of Spindle when shift gear (10ms)	10
63	Braking Time of Spindle when shift gear (10ms)	10
64	Delay time between reset M03/M04 & set M05 (10ms)	0
68	Delay time when feeding axes shift direction	0
69	Mode of Manual Return Home(1:All axis, 0:Selected Axis)	0
80	X/Z axis is Rotating Axis	00000001
81	Mode of Y Axis [0:Rotating Axis, 1: Linear Axis]	1
82	Workpiece coordinate when Y is rotating axis [0 No;1 plan]	1
83	Machine Coordinate when Y is rotating axis [0 No;1 plan]	1
96a	Mode of C Axis [0:Rotating Axis, 1: Linear Axis]	0
96b	Workpiece coordinate when C is rotating axis [0 No;1 plan]	0
96c	Machine Coordinate when C is rotating axis [0 No;1 plan]	0
97a	Mode of Xs Axis [0:Rotating Axis, 1: Linear Axis]	0
97b	Workpiece coordinate when Xs is rotating axis [0 No;1 plan]	0
97c	Machine Coordinate when Xs is rotating axis [0 No;1 plan]	0

98a	Mode of Ys Axis [0:Rotating Axis, 1: Linear Axis]	0
98b	Workpiece coordinate when Ys is rotating axis [0 No;1 plan]	0
98c	Machine Coordinate when Ys is rotating axis [0 No;1 plan]	0
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 [+1:Workpiece Coor +2: Machine Coor]	0
103	Automatically Output(Y/M) for unclamp when 4 th Axis is running	0
104	Detection in position of unclamp[10000+:X,20000+:M,30000+:wait time]	0
110	System Inner Parameter	7
111	Mode of B Axis [0:Rotating Axis, 1: Linear Axis]	1
112	Machine Coordinate when B is rotating axis [0 No;3 plan]	0
113	Name of 5th Axis [0:B, 1:C]	0
200	Pulses Per Revolution of Spindle	4096
200-1	Alarm Range for spindle resolution on Spindle encoder check	0
200-2	Quadruple Pulses Per Revolution of Spindle	0
201	Detect Position Feedback of Spindle(0:No,1:Yes)	1
202	Detecting Angle when SP-Orientation	0.00
203	Allowable error angle when SP-Orientation	0.00
204	SP_Direction when position control mode	0
205	Using Electronic Gear Ratio for Spindle [0:Yes, 1:No]	0
206	Numerator of SP_Electronic Gear Ratio in Low Gear	4096
207	Denominator of SP_Electronic Gear Ratio in Low Gear	36000
208	Numerator of SP_Electronic Gear Ratio in High Gear	4096
209	Denominator of SP_Electronic Gear Ratio in High Gear	360000
210	Coordinate Axis when spindle do tapping	98
210a	Coordinate Axis when Y-axis do tapping	1
210b	Coordinate Axis when X-axis do tapping	1
211	Control Mode of Tapping [2:Following, 3: Interpolation]	3
212	Teeth of SP_Motor (<P413)	1
213	Teeth of SP_Encoder (>P412)	1
214	Follow-Up of A/B/C Axis[7:X, 8:Y, 9:Z]	1
220	Full pulse control channel for 1 st spindle(88:Normal,98:Reverse)	88
221	Pulses per revolution of 1 st spindle (full pulse control)	10000
222	Acceleration/Deceleration of 1 st spindle(full pulse control)(rpm/2s)	500
223	Full pulse control channel for 2 nd spindle(81-88:Normal,91-98:Reverse)	1
224	Pulses per revolution of 2 nd spindle (full pulse control)	10000
225	Acceleration/Deceleration of 2 nd spindle(full pulse control)(rpm/2s)	500
250	Bits of EtherCAT servo motor's encoder	17
323	Nearest way for rotary axis(+4:X, +8:Y, +16:Z, +32:A, +64:B)	0
501	Advanced distance for Z-axis with M123M125(um)[>9/<-9]	0
502	Rough allowance when advance with M123M125(um)[radius]	0
503	Max feed speed when rough processing with M123M125(mm/min)[>9]	0

Note: System inner parameters cannot be changed by user.

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)

4,Max Travel in X_Positive direction (mm)

5,Max Travel in Y_Negative direction (mm)

6,Max Travel in Y_Positive direction (mm)

7,Max Travel in Z_Negative direction (mm)

8,Max Travel in Z_Positive direction (mm)

9,Max Travel in A_Negative direction (mm)

10,Max Travel in A_Positive direction (mm)

11,Max Travel in X_Negative direction (mm)

12,Max Travel in X_Positive direction (mm)

12a,Max Travel in Y_Negative direction (mm)

12b,Max Travel in Y_Positive direction (mm)

12c,Max Travel in Z_Negative direction (mm)

12d,Max Travel in Z_Positive direction (mm)

12e,Max Travel in A_Negative direction (mm)

12f,Max Travel in A_Positive direction (mm)

These parameters are set for max travel in negative direction/in positive direction of feeding axes when soft-limitation is valid, which is based on machine coordinate system.unit: mm.

13,SP_Braking Time (10ms)

It is the braking time of spindle,also time of output M05 when P14=0. Unit: 10ms.

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

It is set for signal mode of SP_Braking. 1: Long signal, 0:short signal. If P14=1,CNC will output M05 always until M3/M4.

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

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

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	0	0	0	0	0	0	0	0	1
Axis	-	-	-	-	YS	XS	C	-	-	B	A	Z	Y	X	-	-

0: means soft-limitation is valid , 1: means invalid. Example: soft limit of X-axis is invalid, P11= 00000100. Suggest all soft limit are valid even if CNC is fixed with hardware limit switches.

D12=1: don't limit when use MOVE code

D14=1: don't limit when ATC

D15=1: Save incremental coordinate even if power off, 0 means not.

16,X_RReverse Backlash Compensation (um)

17,Y_RReverse Backlash Compensation (um)

18,Z_RReverse Backlash Compensation (um)

19,A_RReverse Backlash Compensation (um)

20,A_RReverse Backlash Compensation (um)

20a,B_RReverse Backlash Compensation (um)

20b,C_RReverse Backlash Compensation (um)

20c,Ys_RReverse Backlash Compensation (um)

These are the value of reverse backlash compensation of feeding axes. When moving direction of feeding axis is changed ,system will make compensation with the value automatically. Unit: um

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

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

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

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

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

25a,Direction for C/Xs/Ys axis(+2:C, +4:Xs, +8:Ys)

It is for shift direction of feeding-axes. 1: Direction of feeding axis is same to direction of code; 0: Direction of feeding axis is opposite to direction of code. P25a are set for C/Xs/Ys axes, example we want to shift direction of C-axis & Ys,so P25a=10(=+2+8).

26,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.

27,Numerator of X_Electron Gear (1-999999) 28,Denominator of X_Electron Gear (1-999999)
 29,Numerator of Y_Electron Gear (1-999999) 30,Denominator of Y_Electron Gear (1-999999)
 31,Numerator of Z_Electron Gear (1-999999) 32,Denominator of Z_Electron Gear (1-999999)
 33,Numerator of A_Electron Gear (1-999999) 34,Denominator of A_Electron Gear (1-999999)
 35,Numerator of B_Electron Gear (1-999999) 36,Denominator of B_Electron Gear (1-999999)
 36a,Numerator of C_Electron Gear (1-999999) 36b,Denominator of C_Electron Gear (1-999999)
 36c,Numerator of Xs_Electron Gear (1-999999) 36d,Denominator of Xs_Electron Gear (1-999999)
 36e,Numerator of Ys_Electron Gear (1-999999) 36f,Denominator of Ys_Electron Gear (1-999999)

These are set for electric ratio of feeding axes, which are composed by Numerator & Denominator of Feeding-axis electron gear ratio.(CMR/CMD) Range: 1-999999.

➤ Algorithm of P27-P36f 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.

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

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 needs 5000 pulses, after which the machine tool moves 5mm, then:

$$\text{CMR/CMD} = 5000 / (5 * 1000) = 1 / 1$$

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 needs 5000 pulses, after which the machine tool moves 10mm.

$$\text{CMR/CMD} = 5000 / (10 * 1000) = 1 / 2$$

That is to say, we can set the values as :CMR=1,CMD=2.

Here, the pulse equivalent is 0.002mm.

There is one simple solution, for example, current ratio is 1:1, X0-X10(CNC move 10mm), check real distance that machine tool moves, if machine just move 5mm, set to 10:5, if machine moves 12mm, set to 10:12. also P27=10, P28=12 on Axis parameter.

37, Type of Limit Switch in Positive Direction[0:NO type, 1:NC type]

It is set type of all limit switches in positive direction, which are connected to +L, X10. 0:NO Type, 1: NC Type. If NO type limit switches, please make parallel connections for all in positive direction, if NC type limit switches, please make series connections for all in positive direction, same wiring diagram for limit switches in negative direction.

38, Type of Limit Switch in Negative Direction[0:NO type, 1:NC type]

It is set type of all limit switches in negative direction, which are connected to -L, X09. 0:NO Type, 1: NC Type.

39,Type of Home [D3X;D4(C)Y;D5Z;D6A;D7B;D8C;D9XS;D10YS;0:Switch;1:float Zero]

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

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	0	0	0	0	0	0	0	0	1
Axis	-	-	-	-	-	Ys	Xs	C	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=0000000000001001.

D0&D1 bits are set coordinate vase of Y-axis

40,X_Machine Coordinate of float zero point 41,Y_Machine Coordinate of float zero point

42,Z_Machine Coordinate of float zero point 43,A_Machine Coordinate of float zero point

44,B_Machine Coordinate of float zero point 44a,C_Machine Coordinate of float zero point

44b,Xs_Machine Coordinate of float zero point 44c,Ys_Machine Coordinate of float zero point

It is set the machine coordinate value of feeding-axes based on float zero point. The value is distance between current position of machine tool and float zero point.

45,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.

46,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.

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

It sets the direction 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	-	-	-	-	-	-	Ys	Xs	C	B	A	Z	Y	X	-	-

0:Homing in positive direction, 1:Homing in negative direction, example,if you want let X-axis go home with negative direction,set P47=0000000000000101.

P47_D1: weather detect unclamp/clamp in position of spindle chuck,1:Yes, 0:No.

If P47_D1=1,detect position of clamp/unclamp, when M10,tighten ,CNC will detect clamp position by M12; when M11, loosen, CNC will detect unclamp position by M14.After valid,CNC will do next step.

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

D6=1: Handwheel/Auto status shift automatically.

D7=1: Manual/Auto status shift automatically ;when end of program, CNC will enter manual status automatically, when press Start key on manual status, CNC will enter Auto status automatically.

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

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

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

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

53, Range of Detect Z0 in B axis (unit: 100um)

It is range that system can detect Z0 signal of feeding-axes encoder . unit: 0.1mm

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

54, Offset after homing in X axis (unit: 10um)

55, Offset after homing in Y axis (unit: 10um)

56, Offset after homing in Z axis (unit: 10um)

57, Offset after homing in A axis (unit: 10um)

58, Offset after homing in X axis (unit: 10um)

58-1, Offset after homing in Y axis (unit: 10um)

58-2, Offset after homing in Z axis (unit: 10um)

58-3, Offset after homing in A axis (unit: 10um)

It is offset that feeding-axes after homing done. Move with G00 speed. Unit: 0.01mm.

59, 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.

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

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

61, 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.

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

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

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

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

64, Spindle stop time (unit: 10ms)

It is the delay time between cancel M03/M04 and boot M05. unit: 10ms.

68, Delay time when feeding axes shift direction (ms)

It sets delay time when feeding axes 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
Function	XR	ZR	XM	ZM	XW	ZW	-	-

81, Mode of Y Axis [0: Rotating Axis, 1: Linear Axis]

82, Workpiece Coordinate when Y is rotating axis [0 No; 1 plan]

83, Machine Coordinate when Y is rotating axis [0 No; 1 plan]

P81 is set for the mode of Y-axis, 0: Rotating axis , 1: Linear axis. When Y-axis is set as rotary

axis,if based on workpiece coordinate system,set P82=1; When Y-axis is set as rotary axis,if based on machine coordinate system,set P83=1.

96a,Mode of C Axis[1:Rotating Axis,0:Linear Axis]

96b,Machine Coordinate when C is rotating axis[0 No;1 plan]

96c,Workpiece Coordinate when C is rotating axis[0 No;1 plan]

P96a is set for the mode of C-axis, 0: Rotating axis , 1: Linear axis. When C-axis is set as rotary axis,if based on workpiece coordinate system,set P96b=1; When Y-axis is set as rotary axis,if based on machine coordinate system,set P96c=1.

97a,Mode of XS Axis[1:Rotating Axis,0:Linear Axis]

97b,Machine Coordinate when XS is rotating axis[0 No;1 plan]

97c,Workpiece Coordinate when XS is rotating axis[0 No;1 plan]

P97a is set for the mode of Xs-axis, 0: Rotating axis , 1: Linear axis. When Xs-axis is set as rotary axis,if based on workpiece coordinate system,set P97b=1; When Xs-axis is set as rotary axis,if based on machine coordinate system,set P97c=1.

98a,Mode of YS Axis[1:Rotating Axis,0:Linear Axis]

98b,Machine Coordinate when YS is rotating axis[0 No;1 plan]

98c,Workpiece Coordinate when YS is rotating axis[0 No;1 plan]

P98a is set for the mode of Ys-axis, 0: Rotating axis , 1: Linear axis. When Y-axis is set as rotary axis,if based on workpiece coordinate system,set P98b=1; When Y-axis is set as rotary axis,if based on machine coordinate system,set P98c=1.

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[+1:Workpiece Coor +2: Machine Coor]

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

103,Automatically Output(Y/M) for unclamp when 4th Axis is running

It is set output point(Y) or auxiliary relay(M) for unclamp automatically when 4th axis is running. Normally 4th axis is used for rotary table,which is with brake, we can define output point/auxiliary relay for control released of brake. 10000+Y output address, 20000+M auxiliary relay address. Example: we use Y27 to control & release brake of rotary table,so P103=10027.

104,Detection in position of unclamp[10000+X,20000+M,30000+wait time]

It is set input point & waiting time for detecting in position of unclamp when rotary table is with brake.10000+(X),set input point address, 20000+(M),set auxiliary relay input,30000+, set waiting time,unit is ms. Example: we use X01 as input point for detect position of unclamp,max waiting time is 1.5seconds,also P104=41501=(10001+31500)

110,Inner parameter

Inner parameter of system, cannot be altered.

111,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.

112,Machine Coordinate when B is rotating axis [+1:Workpiece Coor +2: Machine Coor]

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

113,Name of 5th Axis [0:B, 1:C]

It set display of 5th axis,if P113=1,set to C, display & programming with "C", 5th axis will display C. Default is P113=0, set to B , display & programming with "B".

200,Pulses Per Revolution of Spindle

It is pulses per revolution of spindle. Pulses= (Resolution of SP-encoder)*4. default is for 1024ppr encoder,also P200=4096(=1024*4).

200-1,Allow Error of Resolution of SP_Encoder[>10]

It sets resolution of spindle encoder,which is checked by CNC,also Spindle encoder check on Alarm/diagnosis screen.Range is >10

200-2,Pulses Per Revolution of Spindle[1/1:0,Other:>99]

It set pulses of spindle per one revolution(Quadruple),when teeth on encoder & on spindle axle is same(1:1), set P200-2 to 0. if not 1:1,valid range is >99,value must be integer.

201,Detect SP_Position Feedback [0: No,1: Yes,16:AB-Reverse]

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. When P201=16,shift AB phase,also shift direction of counting AB phase signals.

202,Spindle orientation detection angle

It set detection angle ,also SP value after spindle do orientation(M61/M22).

203,Angle measurement error of spindle orientation

It set allow error between SP value after spindle orientation & value of P202.

204,SP_Direction when position control mode[0:Reverse, 1:Normal]

It set direction of spindle axis in position control mode(pulse control).

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

It set whether using electronic gear ratio for spindle, 0:Yes,1:no.

206,Numerator of SP_Electronic Gear Ratio in Low Gear(1-32767)

207,Denominator of SP_Electronic Gear Ratio in Low Gear(1-32767)

208,Numerator of SP_Electronic Gear Ratio in High Gear(1-32767)

209,Denominator of SP_Electronic Gear Ratio in High Gear(1-32767)

These are set for ratio for spindle axis when it is with low/high gear.

210,Coordinate Axis as SP when tapping[91X,92Y,93Z,94A,95B,96C,97Xs,98Ys]

It is set coordinate axis for control spindle axis when interpolate tapping.91:X-axis, 92:Y-axis, 93:Z-axis,94:A-axis,95:B-axis,also 5th axis; 96:C-axis,also 6th axis; 97:Xs-axis, 98:Ys-axis.

210a,Y-axis Interpolation tap SP name[91X,92Y,93Z,94A,95B,96C,97Xs,98Ys]

210b,X-axis Interpolation tap SP name[91X,92Y,93Z,94A,95B,96C,97Xs,98Ys]

These are set coordinate axis for Y-axis/X-axis tapping cycle

211,Control Mode of Tapping [2:Following,3:Interpolation]

It set mode of tapping cycle, 2:tapping with following spindle encoder, 3: interpolate tapping.

212,Teeth of SP_Motor(<P413)

213,Teeth of SP_Encoder(>P412)

These are set for teeth on spindle motor & on spindle encoder.

214,Follow-Up of A/B/C Axis

These parameters are set for follow up axis of A/B/C axis, also set of master-slave axis.

7: A-axis is as slave axis of X-axis 8: A-axis is as slave axis of Y-axis

9: A-axis is as slave axis of Z-axis

17: B-axis is as slave axis of X-axis 18: B-axis is as slave axis of Y-axis

19: B-axis is as slave axis of Z-axis

27: C-axis is as slave axis of X-axis 28: C-axis is as slave axis of Y-axis

29: C-axis is as slave axis of Z-axis

220,Full pulse control channel for 1st spindle(88:Normal,98:Reverse)

It set Ys axis as full pulse control axis for 1st spindle & direction,also use pulse signals to control speed of spindle, ifP220=1, use analog voltage(0-10V) to control speed of 1st spindle.88: Normal direction with Ys axis, 98:set Ys axis as reverse direction for 1st spindle.

221,Pulses per revolution of 1st spindle (full pulse control)

It set pulses per revolution for 1st spindle with full pulse control. Example: if with SZGH spindle servo system,set to 4096.

222,Acceleration/Deceleration of 1st spindle(full pulse control)(rpm/2ms)

It set acceleration/deceleration of 1st spindle with full pulse control,unit: rpm/2ms.

223,Full pulse control channel for 2nd spindle(81-88:Normal,91-98:Reverse)

It set X/Y/Z/A/B/C/Xs/Ys axis as full pulse control axis for 2nd spindle & direction,also use pulse signals to control speed of spindle, ifP220=1, use analog voltage(0-10V) to control speed of 2nd spindle.81~88: Normal direction with X/Y/Z/A/B/C/XS/YS, 91~98: reverse direction for 2nd spindle.

224,Pulses per revolution of 2nd spindle (full pulse control)

It set pulses per revolution for 2nd spindle with full pulse control. Example: if with SZGH spindle servo system,set to 4096.

225,Acceleration/Deceleration of 2nd spindle(full pulse control)(rpm/2s)

It set acceleration/deceleration of 2nd spindle with full pulse control,unit: rpm/2ms.

250,Bits of EtherCAT servo motor's encoder

It set bits of EtherCAT servo motor's encoder.

323,Nearest way for rotary axis(+4:X, +8:Y, +16:Z, +32:A, +64:B)

It sets whether these rotary axes are moving with nearest way, +4: X-axis don't run with nearest way, +8:Y-axis, +16:Z-axis, +32:A-axis, +64:B-axis.

501,Advanced distance for Z-axis with M123M125(um)[>9/<-9]

It sets advanced distance for Z-axis when use M123/M125 follow-up codes.Range:>9/<-9.

502,Rough allowance when advance with M123M125(um)[radius]

It sets rough allowance when advance with M123/M125 code, radius value,unit:um.

503,Max feed speed when rough processing with M123M125(mm/min)

It sets max feeding speed when rough processing with M123/M125 code,unit: mm/min.

518,X-P30 M133 Mode(0:Constant speed,1:Oscillation,2:Fixed length)

518-1,X-P40 M133 length low-16bits 518-2,X-P41 M133 length high-16bits

518-3,X-P46 M133 delay(ms)

528,Y-P30 M133 Mode(0:Constant speed,1:Oscillation,2:Fixed length)

528-1,Y-P40 M133 length low-16bits 528-2,Y-P41 M133 length high-16bits

528-3,Y-P46 M133 delay(ms)

538,Z-P30 M133 Mode(0:Constant speed,1:Oscillation,2:Fixed length)

538-1,Z-P40 M133 length low-16bits 538-2,Z-P41 M133 length high-16bits

538-3,Z-P46 M133 delay(ms)

548,A-P30 M133 Mode(0:Constant speed,1:Oscillation,2:Fixed length)

548-1,A-P40 M133 length low-16bits 548-2,A-P41 M133 length high-16bits

548-3,A-P46 M133 delay(ms)

558,B-P30 M133 Mode(0:Constant speed,1:Oscillation,2:Fixed length)

558-1,B-P40 M133 length low-16bits 558-2,B-P41 M133 length high-16bits

558-3,B-P46 M133 delay(ms)

568,C-P30 M133 Mode(0:Constant speed,1:Oscillation,2:Fixed length)

568-1,C-P40 M133 length low-16bits 568-2,C-P41 M133 length high-16bits

568-3,C-P46 M133 delay(ms)

578,Xs-P30 M133 Mode(0:Constant speed,1:Oscillation,2:Fixed length)

578-1,Xs-P40 M133 length low-16bits 578-2,Xs-P41 M133 length high-16bits

578-3,Xs-P46 M133 delay(ms)

588,Ys-P30 M133 Mode(0:Constant speed,1:Oscillation,2:Fixed length)

588-1,Ys-P40 M133 length low-16bits 588-2,Ys-P41 M133 length high-16bits

588-3,Ys-P46 M133 delay(ms)

8-0,Current drive parameter setting shaft[1--8]

8-1,Modify the current axis parameter password(11:valid,Others:invalid)

8-8,Current loop proportional gain

8-9,Speed loop proportional gain

8-10,Position feedforward gain

8-11,Position loop proportional gain

8-31,Driving alarm overload percentage

8-32,Rated current of motor(0.1A)

8-33,Rated torque of motor(0.1NM)

8-44,Current loop filter constant

8-34,Zero position motor encoder

8-35,Motor pole logarithm

8-48,Driving control parameters(12:Current initialization current)

8-51,Check the encoder alarm [0:Yes,1:No]

8-53,Delay time between power on motor and lock release (ms)

8-54,Delay time for opening switch to power down motor(ms)

8-57,Number of motor coder lines

8-59,Motor stop current locking ratio

8-30,M133 Control mode(0:constant speed,1:reciprocating swing,2:constant length)
8-40,M133 Swing distance (low 16 bits)
8-41,M133 Swing distance (high 16 bits)
8-46,M133 Swing stop time(ms)

Above parameters are remain for developing in the future.

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
8	Management of Tool(0:M06,1:T code,+32768:Tool life management)	1
10	Change tool function[1 Yes, 0 No]	1
11	Go home axis when change tool[3 mean ZX;4 mean ZY;5 mean ZXY;Others means Z]	0
12	Go home reference point when change tool[1,2,3,4]	2
13	Check SP fix position(1 Yes, 0 No)	1
14	SP go to fix position when change tool[1 Yes, 0 No]	1
15	Tools storeroom type[0 Revolve machine hand,1 line tools,2 circle plate]	2
16	Special Tools storeroom[0:Standard , 16: Special Circle]	0
17	Z-axis coordinate when change Circle plate tool storeroom	120
18	Z-axis speed when change Circle plate tool storeroom	2000
19	SP chuck close delay(0.1s)	5
20	SP Chuck loose delay(0.1s)	5
21	Tool cover control delay(ms)	5
22	22,Does lock for Z-axis move & Tools storeroom control(0 mean no,1 mean Yes)	1
23	Does check Z-axis position when Tools storeroom go forward instruction M71 is valid	1
24	Maximum tool number(setup this parameter,must initialize tool table)	0
25	Set up T08 signal[rise "1000+input number",drop "2000+input number"]	1007
26	Set up M61 signal[1000+output number]	1041
27	Set up M63 signal[1000+output number]	1001
28	Set up M71 signal[1000+output number]	1005
29	Set up X42 signal[1000+output number]	1042
30	Set up Y24 signal[1000+output number]	1024
32	Filtering for Position Signal or WAT Signal	1287
70	Servo tools go home speed(degree/min)	0
71	Servo tools go home reverse speed(degree/min)	0
72	Servo tools go home offset(0.001degree)	0
73	Servo change tool speed(degree/min)	0
74	Servo tools axis(1=A; 2=B; 3=C)	0
75	Servo tools acceleration(degree/min/s)	0
76	Servo tools go home max distance(0.001degree)	0
77	Servo tools go home check motor zero(0: is No. 1: is Yes)	0
78	Servo tools revolve one tool pulse	0
79	Servo tools go home current tool No.	0
100	Reference point 1 X(mm)	0
101	Reference point 1 Y(mm)	0
102	Reference point 1 Z(mm)	0
103	Reference point 1 A(mm)	0
104	Reference point 2 X(mm)	0
105	Reference point 2 Y(mm)	0
106	Reference point 2 Z(mm)	50

107	Reference point 2 A(mm)	0
108	Reference point 3 X(mm)	0
109	Reference point 3 Y(mm)	0
110	Reference point 3 Z(mm)	0
111	Reference point 3 A(mm)	0
112	Reference point 4 X(mm)	0
113	Reference point 4 Y(mm)	0
114	Reference point 4 Z(mm)	0
115	Reference point 4 A(mm)	0

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.

8,Management of Tool(0:M06,1:T code,+32768:Tool life management)

It sets management way for tool code,0:use M06 with T code, 1: use T code for change tool directly, +32768, activate function of tool lift management.

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: 1.Tool parameters for tool magazine are exist in CNC system specially for machine center. 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, 88:Yes)	88
4	Lubricate Automatically (0:Yes, 1:No)	1
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	0000010001000100
10	Counting Workpiece Automatically(0:No, 1:Yes)	1
11	Increment of shift block	1
12	System Inner Parameter	16
13	Interlock between Rotation_SP & Chuck(0:No, 1:Yes)	0
14	Coolant key is valid on Auto(0:No, 1:Yes)	1
17	Type of Driver Alarm(ALM)(0:NO type, 1:NC type)	1
17-1	Type of Driver Alarm(X22)[1:NO type,0:NC type]	1
17-2	Braker Output Y16[1:YES,0:NO]	1
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)	1
20	Control Mode of Chuck(0:Single, 1:Double)	0
22	External Switch for Chuck(0:No, 1:Yes)	0
24	Time of Chuck(s)	0.00
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, 18:Yes)	18
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
37-1	OPC Function_ModBus_Station	0
38	Latched for Rapid Key(8:Yes)	1
39	System Inner Parameter	***
40	System Inner Parameter	-88
41	Backup Current Parameters	
42	Recovery Backup Parameters	
50	Run from middle Program ask going last line point[8:Yes,0:No]	
51	Return Safe Point firstly when starting from break point(+4:Z,+8:X)	
52	Machine Coordinate_Safe Point	
120	Direction of Manual Feeding Key	1
140	M77:Machine Coordinate VS Reference point 1	
141	M78:Machine Coordinate VS Reference point 2	
150	X_Reference Point_1(mm)	

151	Y_Reference Point_1(mm)	
152	Z_Reference Point_1(mm)	
153	A_Reference Point_1(mm)	
160	X_Reference Point_2(mm)	
161	Y_Reference Point_2(mm)	
162	Z_Reference Point_2(mm)	
163	A_Reference Point_2(mm)	
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
209a	C_Feedback Allow Error When Stopping (Pulse) [>1]	1
209b	Xs_Feedback Allow Error When Stopping (Pulse) [>1]	1
209c	Ys_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
214a	Numerator of C_Electron Gear for Position Feedback	1
214b	Numerator of Xs_Electron Gear for Position Feedback	1
214c	Numerator of Ys_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
219a	Denominator of C_Electron Gear for Position Feedback	1
219b	Denominator of Xs_Electron Gear for Position Feedback	1
219c	Denominator of Ys_Electron Gear for Position Feedback	1
300	Feeding Axes with Absolute Servo Motor(0:No, 1:Yes)	0000000000000001
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	283
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
324a	Ratio of Gearbox in C-Axis(Absolute)	1
324b	Ratio of Gearbox in Xs-Axis(Absolute)	1
324c	Ratio of Gearbox in Ys-Axis(Absolute)	1
350	Internet[0:OFF,1:ON,8:Auto ON when boot]	0
351	Add._IP	0.0.0.10
352	Add._IP_CNC	0.0.0.10
353	Add._MAC_CNC	0:0:0:1
354	Add._IP_Net Server	0.0.0.0
361	IP address of FTP Server for wifi Hot	0.0.0.0
362	WiFi hot user name	
363	WiFi hot user code	
364	WiFi hot spot[8:Open,Others:Close]	0
365	FTP user name	
366	FTP code	
380	X_Machine Coordinate Value at initial point(ATS)(mm)	0.00
381	Y_Machine Coordinate Value at initial point(ATS)(mm)	0.00
382	Z_Machine Coordinate Value at initial point(ATS)(mm)	0.00
383	Speed in Negative Direction(ATS)(mm/min)	0.00
384	Speed in Positive Direction(ATS)(mm/min)	0.00
385	Z_Workpiece coordinate Value at surface workpiece after ATS(mm)	0.00
386	Positioning Speed(ATS)(mm/min)	0.00
387	Mode of ATS(1:Fixed point,0: Float Point)	0.00
388	Z_Min Machine Coordinate Value(mm)	0.00
389	The Gap value of Z(mm)	0.00
390	Total number of tests for automatic tool(1--5)	0.00
400	Axis name display configuration [xxxxx2]	1
401	Pulse output ports Configuration for Feed Axes[xxxxxxxx0]	1
501	Shift Color Display of Screen(1:No, 8:Yes)	1
591	Define Parameters for Step	
592	Define Parameters for Servo	Servo
600	Control type(4000-4099:EtherCAT,6000:Modbus,6001:1-6axis Modbus, 6002:3axis Modbus)	1
601	EtherCAT Position loop gain coefficient	40
603	EtherCAT Position in position allowable difference (number of motor encoder pulses)	40
610	EtherCAT Electronic gear common denominator	10005

611	EtherCAT X Electronic gear molecule	5000
612	EtherCAT Y Electronic gear molecule	65536
613	EtherCAT Z Electronic gear molecule	65536
614	EtherCAT A Electronic gear molecule	65536
615	EtherCAT B Electronic gear molecule	65536
616	EtherCAT C Electronic gear molecule	65536
617	EtherCAT Xs Electronic gear molecule	65536
618	EtherCAT Ys Electronic gear molecule	65536
620	EtherCAT Station number setting [> 10]	12345678
621	EtherCAT Servo drive type [> 1]	11111111
622	EtherCAT J1 Collision threshold	1000
623	EtherCAT J2 Collision threshold	1000
624	EtherCAT J3 Collision threshold	1000
625	EtherCAT J4 Collision threshold	1000
626	EtherCAT J5 Collision threshold	1000
627	EtherCAT J6 Collision threshold	1000
900	Display User-define Dialog Box[1:No, 4:Yes]	8192
901	Homing Sequence of Axis(5~9bits)	1
902	Inner Parameter[2:ON]	0
903	Inner Parameter[2:SD Card,+16:5th Axis Follow,+32:4th Axis Follow]	19
904	Inner Parameter[4:File Decrypt,8:Encryption]	0
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
994	CNC controller serial number	
995	Machine tool serial number	
996	User-defined serial number	

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 be used as sp-rate & feed-rate,only off/X/Y/Z/A/B & *1/*10/*100(also P1&P2=0 in Axis parameter if use CN11 as rate knob input).

3,Using Interface Switch on Panel(0: No, 88:Yes,)

It sets whether system use interface switch(3 position switch/Normal/F-Halt/FS-halt) on operational panel. 0:No, don't use; 88:Yes,use on Manual/Auto both,99:Yes, use on Auto

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 running time .

5,Time of Lubrication (10ms)

It sets the time of lubricate automatically , also time of outputting M32/Y08. 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, X29.

2. After set P7=1&M12 is valid,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,details 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	1	1	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;0:Yes

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

D7: Null; default value is 0,which cannot be altered.

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

D10: 1:Auto Sequence for block when programming;

D11: 1:Analog of 1st spindle outputs through both analog channel(AV1/AV2);2nd SP invalid.

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 Workpiece 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 number 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, semi-interlock; 8:fully interlock, 16:super interlock, 32:interlock when run program automatically, 56:Compulsion interlock. only when chuck is clamping, and then spindle can rotate; when spindle is rotating, Chuck cannot unclamping. Suggest set to interlock for safe.

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.

17, Type of Driver Alarm(X20)(1:NO type, 0:NC type)

17-1, Type of Driver Alarm(X22)[1:NO type, 0:NC type]

It sets the type of driver alarm. ALM, X20 is input point for alarm signals of XYZA drivers normally, X22 is for BCXSYS drivers. 0:NC type; 1:NO type.

17-2, Braker Output Y16[1:YES, 0:NO]

It is set whether output Y16 for control Braker when servo drivers are normal, 1:Yes, 0:No.

18, Type of Spindle Alarm(X23/ALM1)(0:NO type, 1:NC type)

It sets the type of spindle alarm. ALM1, X23 input port. 0:NO type; 1:NC type.

19, Type of Machine Alarm(X24/ALM2)(0:NO type, 1:NC type)

It sets the type of machine_tool alarm. ALM2, X24 input port. 0:NO type; 1:NC type.

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

It sets control mode of chuck, 0:Single control signal ; 1:Double control signal(M10/M71).

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

P20=1, two output points for Chuck, M10: output M71(Y05 port) to clamp chuck; M11: output M10(Y10) to unclamp chuck.

P40_D1 on Axis parameter sets if detect position of chuck unclamp/clamp tool.

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, X31 Plug.

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

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. Unit:s.

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 PIN5_CN11 Plug. 0: NO type switch; 1:NC type switch for 2nd emergency stop.(E-Stop on Handheld box/external panel). Suggest use NC type E-top button.

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.When CNC is in running condition,output M69, Y04 port of IO board; When CNC is in pause, output M65,Y02 port on IO board.

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 on CNC. 0:No, don't output condition of Alarm; 1:Yes, When CNC is alarming,Output M67,Y03 pin on IO board.

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. 32: Output points & on PLC both are valid when control in diagnosis ; 64: Output point is valid, output on PLC is invalid.

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, 18:Yes)

It sets if use high-speed inner PMC for IOs. 0:No,don't use PMC; 18: Yes,use High-Speed PMC.28:Use Super High-speed.

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.

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. CNC will display Fast-M when Rapid mode.

39&40, parameters are set for decrypt/encrypt CNC controller

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.*

50,Run from middle Program ask going last line point[8:Yes,0:No]

It sets whether CNC moves to end point of last block when start from middle line,8:Yes, 0:No.

51,Return Safe Point firstly when starting from break point(+4:Z,+8:X,+16:Feed Hold)

It sets whether CNC go to safe point firstly when P50=8.+4:Z-axis go to safe point; +8:X-axis; +16: feed hold.

52,Machine Coordinate_Safe Point

It sets machine coordinate value of safe point after set of P51.

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;

140,M77:Machine Coordinate VS Reference point 1(1001/2001/3001/4001:X/Y/Z/A)

It sets whether M77 is valid after compare axis machine coordinate value & coordinate value of 1st reference point, 1001: compare axis is X-axis, 2001:Y-axis,3001:Z-axis, 4001:A-axis. For example: P140=1001, if machine coordinate value is less than P150(X_reference point_1),X-axis value of 1st reference point , M77 is valid.

141,M78:Machine Coordinate VS Reference point 2(1002/2002/3002/4002:X/Y/Z/A)

It sets whether M78 is valid after compare axis machine coordinate value & coordinate value of 2nd reference point, 1002: compare axis is X-axis, 2002:Y-axis,3002:Z-axis, 4002:A-axis. When machine coordinate value is less than coordinate value of 2nd reference point,M78 is valid.

150,X_Reference Point_1(mm) 151,Y_Reference Point_1(mm)

152,Z_Reference Point_1(mm) 153,A_Reference Point_1(mm)

These are XYZA coordinate value for 1st reference point

160,X_Reference Point_2(mm) 161,Y_Reference Point_2(mm)

162,Z_Reference Point_2(mm) 163,A_Reference Point_2(mm)

These are XYZA coordinate value for 2nd reference point

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 lead 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 L***M***,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 lead 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 L***M***, 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 Motor(0:No, 1:Yes)

Bit	D15	D14	D13	D12	D11	D10	D9	D8	D7	D6	D5	D4	D3	D2	D1
Value	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
Func	-	-	-	-	-	-	YS	XS	C	B	A	Z	Y	X	-

It is bit parameter, and sets if each feeding axes is with absolute servo motor or not. 0 means axis motors aren't absolute servo motor; 1 means yes, axes motors are absolute servo motor.

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 485+/485- ports on IO board, communication protocol is as follow:

1) Station Address of RS485 on driver: 1:X axis driver; 2:Y axis driver; 3:Z axis driver; 4:A axis driver; 5:B axis driver; 6:C axis driver; 7:Xs axis driver; 8: Ys axis driver.

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}

◆ **Parameters configuration for Delta absolute servo system**

1. Parameter set on CNC controller:

P301=102, corresponding to P0-51, communication address: 0x66.

P302=104, corresponding to P0-52, communication address: 0x68.

P302=-98, corresponding to P0-49, negative number of communication address: 0x64.

P304/P305/P306/P307/P308, Pulses per revolution of X/Y/Z/A/B/C/Xs/Ys axis: set to 1280000.

2. Parameter set on Delta driver:

P3-00: Station number, 1/2/3/4/5/6/7/8, corresponding to X/Y/Z/A/B/C/Xs/Ys axis.

P3-01: set to 2 ; P3-02: set to 8 ; P3-03: set to 0 ; P3-04: set to 0 ; P3-05: set to 1 ;

P3-07: set to 0 ; P2-69: set to 1 ; P2-70: set to 2 .

◆ **Parameters configuration for Maxsine absolute servo system**

1. Parameter set on CNC controller:

P304/P305/P306/P307/P308, Pulses per revolution of X/Y/Z/A/B/C/Xs/Ys axis: set to 65536.

When pitch of ball screw isn't 10mm, user can set electric ratio on Maxsine driver by P029&P030 (& P031, P032, P033). Pitch of ball screw (P309/P310/P311/P312/P313) are set according to real pitch of ball screw, ratio set for position feedback also is same.

2. Parameter set on Delta driver:

P300: Station number, 1/2/3/4/5/6/7/8, corresponding to X/Y/Z/A/B/C/Xs/Ys axis.

P304: 0: normal mode, 1: Modbus mode, set to 1

P306: set to 2;

P307: 0: incremental position; 1: absolute position; set to 1.

P027/P028: Set for electric ratio, product of P027&P028 is resolution of encoder,ensure control accuracy, for 17bits encoder, set P027 to 8192, P028=16.

After set above parameters, please save with E-set,and reboot drivers.

If fix with Multi-turns encoder&battery, set P90 to 1,and initialize with Fn36.

Threading/Tapping processing: CNC only can processing tapping/threading when spindle speed is within 1000rpm with default parameters. Increase gains of servo driver, CNC can do tapping when spindle speed is within 2000rpm, parameters set as following:

P005: 1 st speed loop gain;	P009: 1 st position loop gain;
X-axis: increase P005 from 40 to 80(Hz)	Increase P009 from 40 to 80(1/s)
Y-axis: increase P005 from 40 to 80(Hz)	Increase P009 from 40 to 80(1/s)
Z-axis: increase P005 from 40 to 100(Hz)	Increase P009 from 40 to 100(1/s)

304,Pulses of One-Revolution in X-Axis

305,Pulses of One-Revolution in Y-Axis

306,Pulses of One-Revolution in Z-Axis

307,Pulses of One-Revolution in A-Axis

308,Pulses of One-Revolution in B-Axis

308a,Pulses of One-Revolution in C-Axis

308b,Pulses of One-Revolution in Xs-Axis

308c,Pulses of One-Revolution in Ys-Axis

It sets pulses of one-revolution in X-axis, which is configured with absolute encoder. The pulses of 17bits absolute encoder is 131072. P304/P305/P306/P307/P308/P308a/P308b/P308c stand for X/Y/Z/A/B /C/Xs/Ys-axis.

309,Pitch of ball screw in X-Axis(nm)

310,Pitch of ball screw in Y-Axis(nm)

311,Pitch of ball screw in Z-Axis(nm)

312,Pitch of ball screw in A-Axis(nm)

313,Pitch of ball screw in B-Axis(nm)

313a,Pitch of ball screw in C-Axis(nm)

313b,Pitch of ball screw in Xs-Axis(nm)

313c,Pitch of ball screw in Ys-Axis(nm)

It sets pitch of ball screw in feeding-axis, which is configured with absolute encoder ,also distance per revolution of ball screw,just for communication. Minus means count's direction is negative. When read data of absolute encoder is too much differ than before,after debug absolute communication twice,try to set to opposite number .

Example: Pitch of ball screw in X-axis is 4mm, so P309=4000000 or -4000000. P309/P310 /P311/P312/P313/P313a/P313b/P313c stands for X/Y/Z/A/B/C/Xs/Ys-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

318a,Offset of Multi-turns in C-axis Absolute Encoder

318b,Offset of Multi-turns in Xs-axis Absolute Encoder

318c,Offset of Multi-turns in Ys-axis Absolute Encoder

It sets Multi-turns offset of X-axis,in order to solve when the values of Multi-turns or

coordinate value is too big ,and overflow etc. problems. set to 'E' for clear value to 0 . P314/ P315/ P316 /P317/P318/P318a/P318b/P318c stand for X/Y/Z/A/B/C/Xs/Ys-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, 231:Load.

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) 324a,Ratio of Gearbox in C-Axis(Absolute)

324b,Ratio of Gearbox in Xs-Axis(Absolute) 324c,Ratio of Gearbox in Ys-Axis(Absolute)

It is ratio of gearbox in feeding-axis,which is configured with absolute encoder. Example: X:Pitch is 4mm, ratio of gearbox is 3:1, so P309=4000000,P320=3. P320/P321/P322/P323/P324/P324a/P324b/P324c stands for X/Y/Z/A/B/C/Xs/Ys axis.

350,Internet[0:OFF,1:ON,8:Auto ON when boot]

It sets whether turn on Internet function, 0: turn off internet function, set to 1: yes, turn on internet function, 8: Automatically turn on internet function after boot.

351,Add. _IP

It sets address of IP, set randomly,just ensure front 3 data are same in local area network. Example: 192.168.1.1.

352,Add. _IP _CNC

It sets address of IP for this CNC system, set randomly,just ensure front 3 data are same in local area network. Example: 192.168.1.103.

353,Add. _MAC _CNC

It sets MAC address of this CNC system, set randomly, just ensure different address for devices in current local area network. Example: 255.255.255.1

354,Add. _IP _Net Server

It sets IP address of net server,also IP address of our computer. It is better that ensure front 3 number are same, example: 192.168.1.13.

USB-Network-WiFi function: Model for wireless USB Network should be EP-N8508GS or BL-LW05-AR5, both are okay,the other model cannot be supported.Set as following:

1.Fix ES-File Browser on your Mobile phone;

2.Open Remote Manager,input IP address to P361 on Other parameter.

If without WIFI signal,we can use WIFI hotspot for set local area network, steps:

1) Open & setup WiFi hotspot on Your Android Mobile phone.

2) Input User account & password to P362&P363 on Other parameter

3) Exit remote manager,and open remote manager again,and input IP address to P361.

Steps of entering Folders of FTP server on CNC system:

1. Press “N” key to enter on Program menu screen.(Shift+N/Rapid+N)
2. After enter folder of FTP server, press N key as steps, or press F6 to exit folder of FTP.

Advantage of FTP folder manager: User can manager files/folders on PC or Mobile phone on CNC system freely. User can anonymous login in FTP servers (don't set P365&P366),and also can login in FTP server with FTP user name & code(password).

Note: when CNC System use WiFi function,CNC only upload/download files with android phone(priority); if CNC system use network, only can upload/download files with PC.

361,IP address of FTP Server for WiFi Hot [Input based on the value of the FTP server in the phone,As:192.168.2.206]

362,WiFi hot user name

363,WiFi hot user code

364,WiFi hot spot[8:Open,Others:Close]

365,FTP user name

366,FTP code

380,X_Machine Coordinate Value at initial point(ATS)(mm)

381,Y_Machine Coordinate Value at initial point(ATS)(mm)

382,Z_Machine Coordinate Value at initial point(ATS)(mm)

383,Speed in Negative Direction(ATS)(mm/min)

384,Speed in Positive Direction(ATS)(mm/min)

385,Z_Workpiece coordinate Value at surface workpiece after ATS(mm)

386,Positioning Speed(ATS)(mm/min)

387,Mode of ATS(1:Fixed point,0: Float Point)

388,Z_Min Machine Coordinate Value(mm)

389,The Gap value of Z(mm)

390,Total number of tests for automatic tool(1--5)

These parameters are set for Automatic tool setter with tool setter/probe, please check Chapter4.7 on Programming book.

400,Axis name display configuration [xxxxx2]

(123456 express XYZABC,Example:123462 shows XYZAC,124502 shows XYAB)

This parameter is just for inner use during debug.

401,Pulse output ports Configuration for Feed Axes[xxxxxxxxx0]

It sets pulse signals output ports for feeding axes,12345678 stands for X/Y/Z/A/B/C/Xs/Ys Example: 123456780, for Example: 123478560, Xs/Ys/B/C axes are output from B/C/Xs/Ys pulse output port.

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.

591,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.

592,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.

600,Control type(4000-4099:EtherCAT,6000:Modbus,6001:1-6axis Modbus,6002:3axis Modbus)

It sets control type for modbus, 4000-4099: configured with EtherCAT servo, 6000: Modbus for all axes; 6001: 1st ~6th axis configured with Modbus servo, 6003:1st ~ 3rd axis configure with Modbus servo. If normal servo system,set to 1.Default is 4003

601,EtherCAT Position loop gain coefficient

602,EtherCAT Thread machining position ring gain coefficient

603,EtherCAT Position in position allowable difference (number of motor encoder pulses)

610,EtherCAT Electronic gear common denominator

611,EtherCAT X Electronic gear molecule

612,EtherCAT Y Electronic gear molecule

613,EtherCAT Z Electronic gear molecule

614,EtherCAT A Electronic gear molecule

615,EtherCAT B Electronic gear molecule

616,EtherCAT C Electronic gear molecule

617,EtherCAT Xs Electronic gear molecule

618,EtherCAT Ys Electronic gear molecule

620,EtherCAT Station number setting [> 10]

It sets for station number for EtherCAT modbus, default is 12345678

621,EtherCAT Servo drive type [> 1]

It sets type of EtherCAT servo driver, if configured with SZGH-SD-EtherCAT series servo driver, set to 77777776.If other model,please contact with us or SZGH agent.

622,EtherCAT J1 Collision threshold

623,EtherCAT J2 Collision threshold

624,EtherCAT J3 Collision threshold

625,EtherCAT J4 Collision threshold

626,EtherCAT J5 Collision threshold

627,EtherCAT J6 Collision threshold

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.4108: Enter with page, +256: US0~54 are user coordinate, +8192: Shift between Coolant key & Rapid key

901,Homing Sequence of Axis(5~9bits)

It sets homing sequence of each axis.Value is 5~9bits.D0 bit is 0. 1:X, 2:C(Y), 3:Z, 4:A, 5:B,

6:C, 7:Xs, 8:Ys. Last bit is 0. 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.



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
1-5	B Workpiece Coordinate (G54-G59)	0.000
1-6	C Workpiece Coordinate (G54-G59)	0.000
1-7	Xs Workpiece Coordinate (G54-G59)	0.000
1-8	Ys Workpiece Coordinate (G54-G59)	0.000
2-0	Current Workpiece Coordinate Set [G54.1-G54.48]	1
2-1	X Workpiece Coordinate (G54.1-G54.48)	0.000
2-2	Y Workpiece Coordinate (G54.1-G54.48)	0.000
2-3	Z Workpiece Coordinate (G54.1-G54.48)	0.000
2-4	A Workpiece Coordinate (G54.1-G54.48)	0.000
2-5	B Workpiece Coordinate (G54.1-G54.48)	0.000
2-6	C Workpiece Coordinate (G54.1-G54.48)	0.000
2-7	Xs Workpiece Coordinate (G54.1-G54.48)	0.000
2-8	Ys Workpiece Coordinate (G54.1-G54.48)	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)

1-5 B Workpiece Coordinate (G54-G59)

1-6 C Workpiece Coordinate (G54-G59)

1-7 Xs Workpiece Coordinate (G54-G59)

1-8 Ys 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-G54.48]

It is for select current workpiece coordinate from G54.1 to G54.48

2-1 X Workpiece Coordinate (G54.1-G54.48)

2-2 Y Workpiece Coordinate (G54.1-G54.48)

2-3 Z Workpiece Coordinate (G54.1-G54.48)

2-4 A Workpiece Coordinate (G54.1-G54.48)

2-5 B Workpiece Coordinate (G54.1-G54.48)

2-6 C Workpiece Coordinate (G54.1-G54.48)

2-7 Xs Workpiece Coordinate (G54.1-G54.48)

2-8 Ys Workpiece Coordinate (G54.1-G54.48)

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 "F5-MDI" key, select corresponding workpiece coordinate system (G54-G59). or press F8 on main screen to choose workpiece coordinate system

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 "F7", press "X" key and 'Enter', insert the measured value, and 'Enter'.

d). Press "F7", press "Z" key and 'Enter', insert the measured value, and 'Enter'.

e). Press "F7", press "Y" key and 'Enter', insert the measured value, and 'Enter'.

f). Press "F7", press "A" key and 'Enter', insert the measured value, and 'Enter'.

Done well now. Enter different workpiece coordinate system, it will show the corresponding value, which also is offset value between workpiece coordinate system and machine coordinate system (G53).

Axis Coordinate value set for all coordinate system:

1) input A###, current axis coordinate offset for all coordinate system, Example: input A12.5 on P1-2 (G54-Y-axis), Y-axis of all coordinate system will offset 12.5mm.

2) Input E, clear to 0; input E###, means input absolute offset value.

3) Input EA, clear all coordinate value of current axis; input EA###, means input absolute offset value to all coordinate system for current axis.

4) On X-axis of anyone coordinate system when parameter set, input EALL, clear all coordinate coordinate value to 0.

Note: There are 54 pcs of workpiece coordinate system: G54~G59 & G54.1~G54.48.

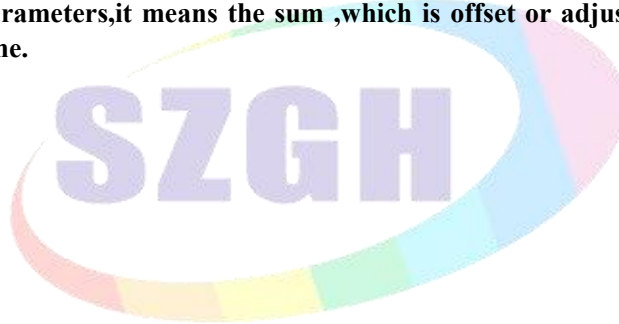
2.6.2 How to adjust the offset value for workpiece coordinate system after set well?

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" aren'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

Note:please must remember new password after alter password.



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.

Man Con		N00000		2017-07-04 18:01																															
Press T Key Length make tool base on mainfac				Program SZGH																															
<table border="1"> <tr><td>T01</td><td>R:</td><td>0.000</td></tr> <tr><td>T02</td><td>R:</td><td>0.000</td></tr> <tr><td>T03</td><td>R:</td><td>0.000</td></tr> <tr><td>T04</td><td>R:</td><td>0.000</td></tr> <tr><td>T05</td><td>R:</td><td>0.000</td></tr> <tr><td>T06</td><td>R:</td><td>0.000</td></tr> <tr><td>T07</td><td>R:</td><td>0.000</td></tr> <tr><td>T08</td><td>R:</td><td>0.000</td></tr> <tr><td>T09</td><td>R:</td><td>0.000</td></tr> <tr><td>T10</td><td>R:</td><td>0.000</td></tr> </table>				T01	R:	0.000	T02	R:	0.000	T03	R:	0.000	T04	R:	0.000	T05	R:	0.000	T06	R:	0.000	T07	R:	0.000	T08	R:	0.000	T09	R:	0.000	T10	R:	0.000	Instruction code G53 T01H0D0	
T01	R:	0.000																																	
T02	R:	0.000																																	
T03	R:	0.000																																	
T04	R:	0.000																																	
T05	R:	0.000																																	
T06	R:	0.000																																	
T07	R:	0.000																																	
T08	R:	0.000																																	
T09	R:	0.000																																	
T10	R:	0.000																																	
				Machine Status																															
				M05 M09 M10 M78 M33 M41																															
				G00 X100% F100 X100% S0 X 5% SP000																															
				Machine Coord																															
				X 0.000 A 0.000 Y 0.000 B 0.000 Z 0.000																															
				PartTime 0: 0																															
				PartNo 0																															
				SPrpm 0																															
No Alarm																																			
F1 Radius F2 Length F3 ACLEA F4 CLEAR F6 ToolSeat F7 Set F8 CANCEL																																			

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.

Man Con		N00000		2017-07-04 18:01																																																					
Press T Key Length make tool base on mainfac				Program SZGH																																																					
<table border="1"> <tr><td>T01</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>T02</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>T03</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>T04</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>T05</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>T06</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>T07</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>T08</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>T09</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>T10</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>H11</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>H12</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> <tr><td>H13</td><td>H:</td><td>0.000</td><td>[0.000]</td></tr> </table>				T01	H:	0.000	[0.000]	T02	H:	0.000	[0.000]	T03	H:	0.000	[0.000]	T04	H:	0.000	[0.000]	T05	H:	0.000	[0.000]	T06	H:	0.000	[0.000]	T07	H:	0.000	[0.000]	T08	H:	0.000	[0.000]	T09	H:	0.000	[0.000]	T10	H:	0.000	[0.000]	H11	H:	0.000	[0.000]	H12	H:	0.000	[0.000]	H13	H:	0.000	[0.000]	Instruction code G53 T01H0D0	
T01	H:	0.000	[0.000]																																																						
T02	H:	0.000	[0.000]																																																						
T03	H:	0.000	[0.000]																																																						
T04	H:	0.000	[0.000]																																																						
T05	H:	0.000	[0.000]																																																						
T06	H:	0.000	[0.000]																																																						
T07	H:	0.000	[0.000]																																																						
T08	H:	0.000	[0.000]																																																						
T09	H:	0.000	[0.000]																																																						
T10	H:	0.000	[0.000]																																																						
H11	H:	0.000	[0.000]																																																						
H12	H:	0.000	[0.000]																																																						
H13	H:	0.000	[0.000]																																																						
				Machine Status																																																					
				M05 M09 M10 M78 M33 M41																																																					
				G00 X100% F100 X100% S0 X 5% SP000																																																					
				Machine Coord																																																					
				X 0.000 A 0.000 Y 0.000 B 0.000 Z 0.000																																																					
				PartTime 0: 0																																																					
				PartNo 0																																																					
				SPrpm 0																																																					
No Alarm																																																									
F1 Radius F2 Length F3 ACLEA F4 CLEAR F5 SetTool F6 ToolSeat F7 Set F8 CANCEL																																																									

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 Tool tip to tool base point, Press T key on main screen to set tool base, input OK for confirm, and press Enter key for confirm. Normally we take surface of working table/workpiece as tool base

2) Move tool tip to tool base point, or a position where is easy to measure coordinate of tools

3) Press "↑ ↓" to move cursor to corresponding tool number

4) Press "F5-SetTool" key and CNC system will pop-up dialog box, press Enter key.

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: length offset can be negative or positive value, radius offset only can be positive number.

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.

Man Con

N00000

2017-07-04 18:02

Press T Key Length make tool base on mainfac

ToolSeat 00	T01
ToolSeat 01	T01
ToolSeat 02	T02
ToolSeat 03	T03
ToolSeat 04	T04
ToolSeat 05	T05
ToolSeat 06	T06
ToolSeat 07	T07
ToolSeat 08	T08
ToolSeat 09	T09
ToolSeat 10	T10

Program

SZGH

Instruction code

G53
T01H0D0

Machine Status

M05 M09 M10
M78 M33 M41
G00 X100%
F100 X100%
S0 X 5% SP000

Machine Coord

X 0.000 A 0.000
Y 0.000 B 0.000
Z 0.000

PartTime

0: 0

PartNo

0

SPrpm

0

No Alarm

F1 INI

F8 CANCEL

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.

Input tool total count:

8

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 in Parameter screen.

ManCon

N00000

2017-07-0418:02

Basic Parameter

1, Standby: 0

2, Neg-point: 3

3, Pos-point: 2

4, multiple: 1.000

5, distance(um): 80000

NoCoor(mm)Value(um)

1)<160.000> 0

2)<80.000> 0

3)<0.000> 0

4)<-80.000> 0

5)<-160.000> 0

6)<-240.000> 0

ProgramSZGH

Instruction code

G53

T01H0D0

Machine Status

M05M09M10

M78M33M41

G00X100%

F100X100%

S0X100%SP000

Machine Coor

X0.000A0.000

Y0.000B0.000

Z0.000

PartTime0: 0

PartNo0

SPrpm0

No Alarm

F1X-axis

F2Y-axis

F3Z-axis

F4A-axis

F5B-axis

F6ChePro

F7CLEAR

F8CANCEL

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.

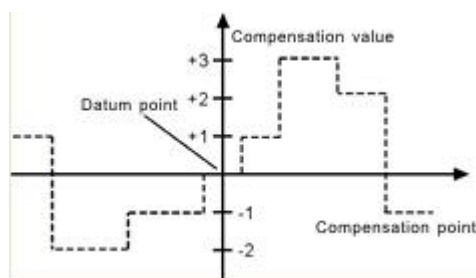
4. Multiple.

It is set for rate of compensation, also $\text{actual value} = \text{set value} * \text{multiple}$.

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. Coord(mm) Value(um)):

In the area of setting compensation value, it will show the value of compensation and every axis' error compensation point of screw pitch. Press " $\uparrow \downarrow$ 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 "F6-ChePro" 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

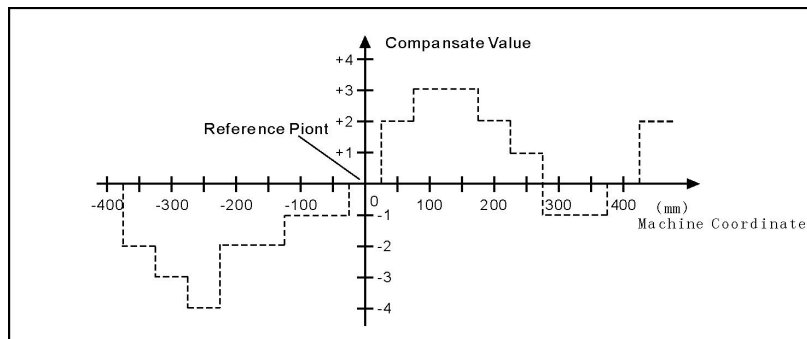
3) Multiple: 1

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

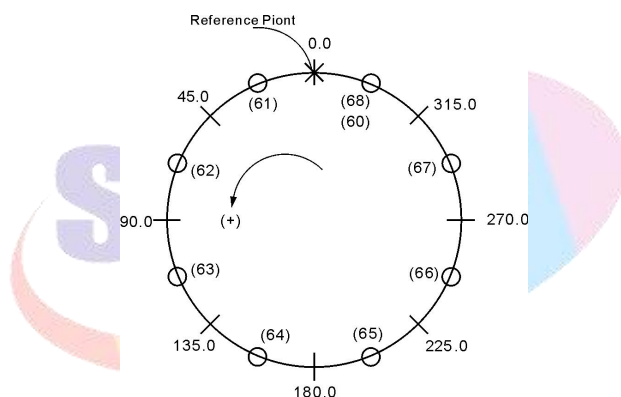
The contrasted chart of compensation points and value as follows:



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:

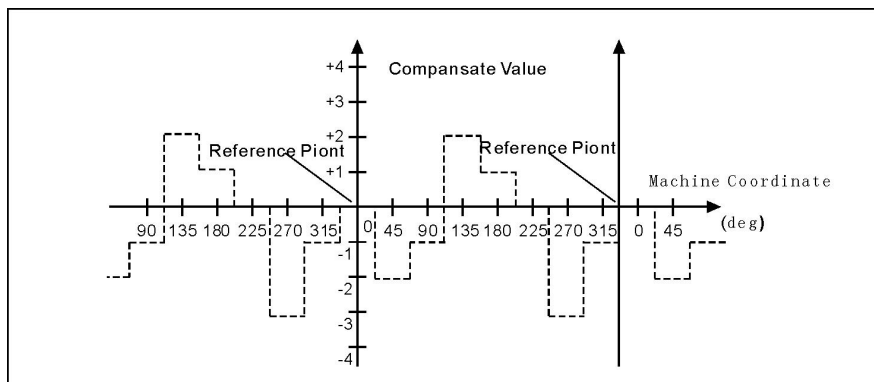
- 1) Backward checking points: 0
- 2) Forward checking points: 8
- 3) Multiple: 1
- 4) Distance(um): 45000



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, unwound 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, weak 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

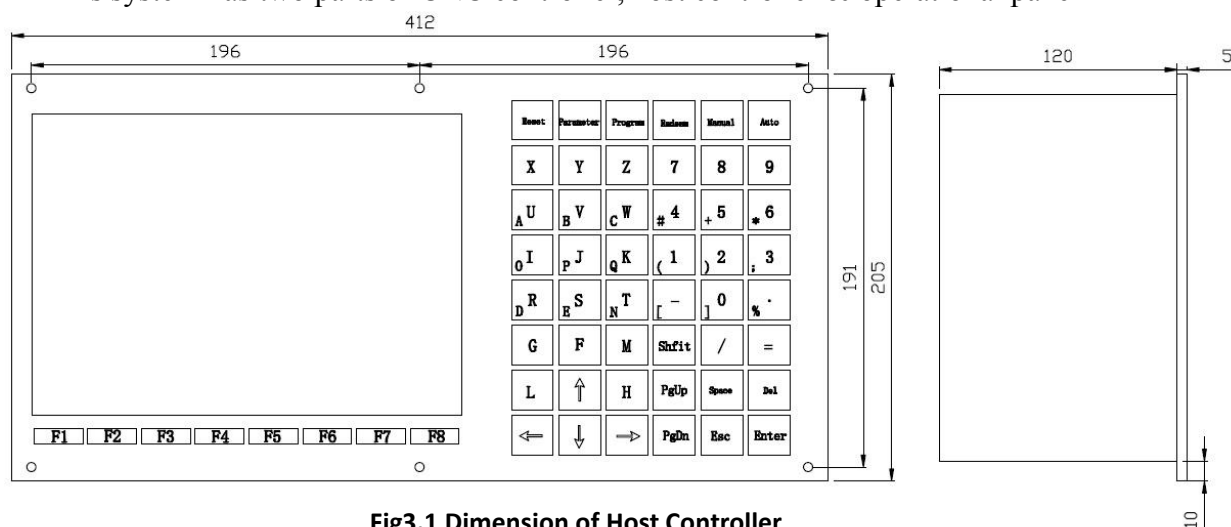


Fig3.1 Dimension of Host Controller

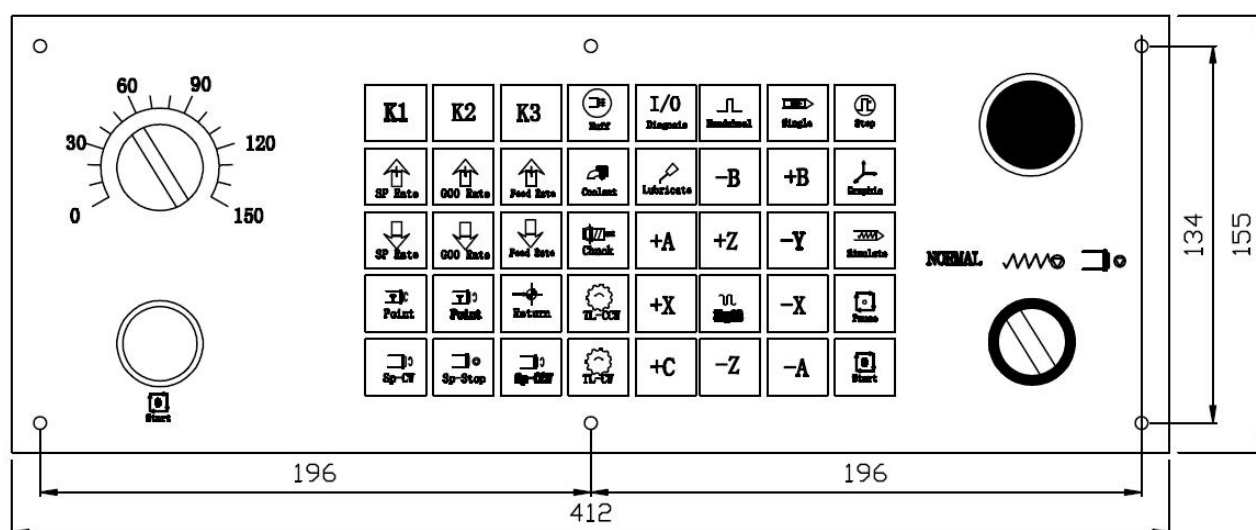
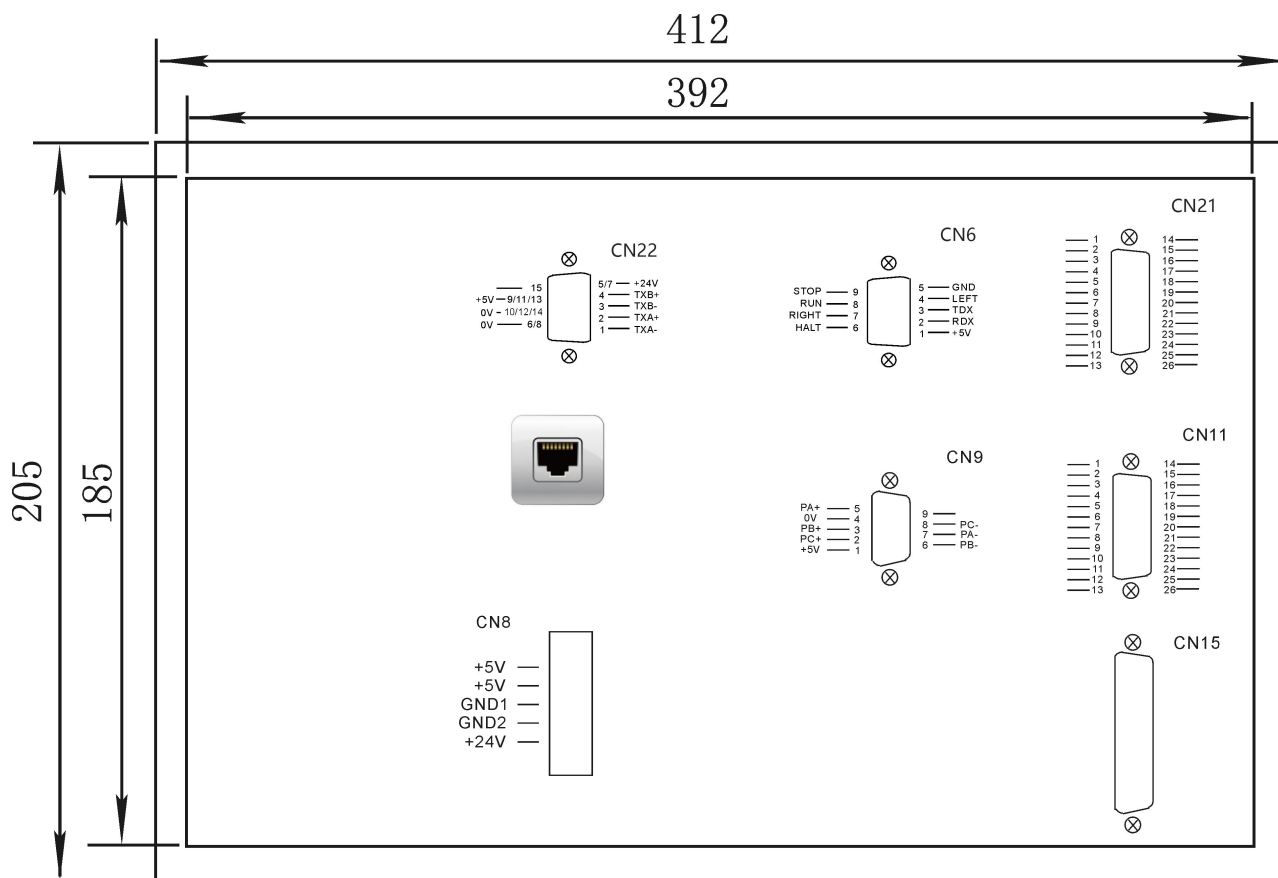


Fig3.2 Dimension of A type & B type Operational Panel

3.3 System Rear View

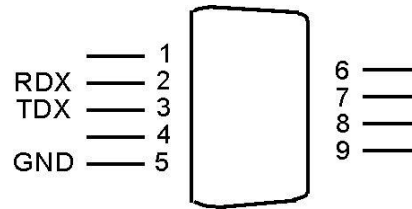


Attention: switching power supply *L*, *N* must be connected to AC 220V, current 0.5A through isolation transformer.

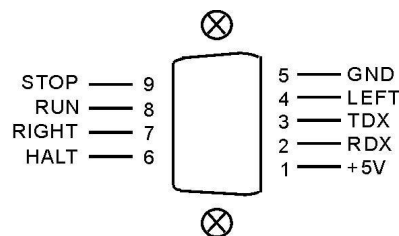
3.4 Interface Connection Graph

3.4.1 Communication Socket (Female/DB9)

1) Socket in the front of Controller



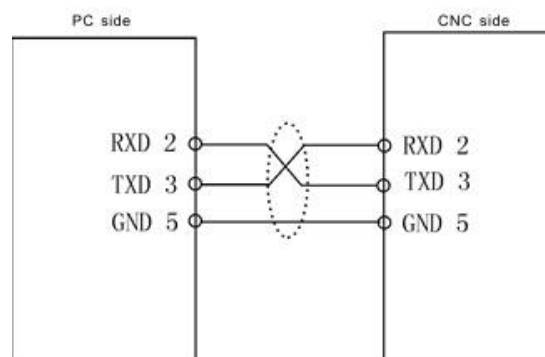
Communication signal with Female socket DB9				
signal	pin	I/O	function	Valid
0V	5	OUT	The ground of signal	0V
RXD	2	IN	The received data signal	
TXD	3	OUT	The transmission of data signal	



2) Socket in the rear of Controller(CN6)

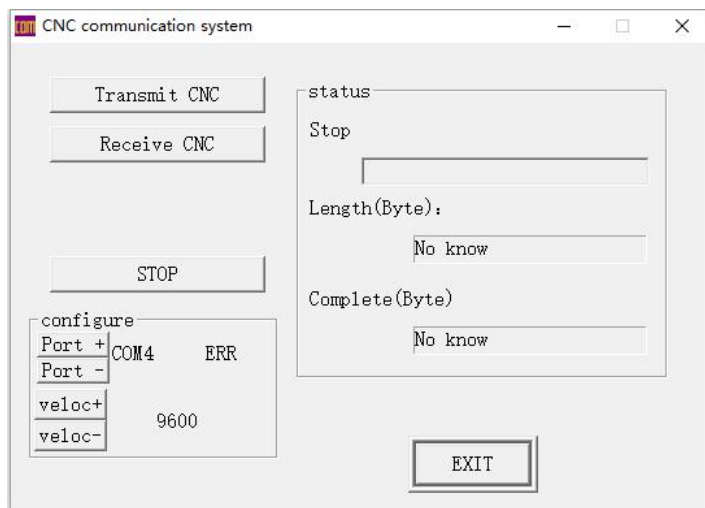
Communication signal with Female socket DB9				
signal	pin	I/O	function	Valid
0V	5	OUT	Ground of signal	0V
RXD	2	IN	Received data signal-RS232	
TXD	3	OUT	Transmission of data signal-RS232	
+5V	1	IN	+5V power	0V
LEFT	4	IN	Left side input-Interference Switch	0V
HALT	6	IN	Pause Input	0V
RIGHT	7	IN	Right side input-Interference Switch	0V
RUN	8	IN	Run Input	0V
STOP	9	IN	Emergency-stop Input	0V

The signal of communication socket connect to PC:



When program is edited by PC ,which should be text files,and could be edit by Notepad or Word pad.

Note:if use USB port of PC,please use one RS232<->USB cable .



SZGHCNCCS software

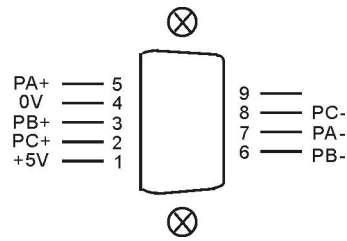
Note: 1. Connect to external PC with data communication, must be equipped with our special communication software, which is "SZGHCNCCS" software.

P37 in Other parameter is set rate of CNC system.(P37=6, rate is 115200)

2.Communication line must adopt the shielded twisted pair cable, length shall not exceed 10m.



3.4.2 Spindle Encoder Socket (Female/DB9)

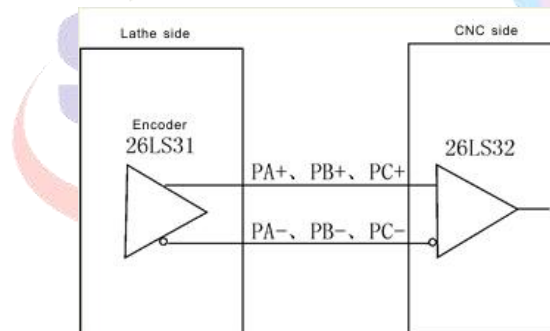


CN9 SP Encoder signal with Female socket DB9				
Signal	Pin	I/O	Function	Valid
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	A Phase Positive signal	5V
PA-	7	IN	A Phase Negative signal	
PB+	3	IN	B Phase Positive signal	5V
PB-	6	IN	B Phase Negative signal	
PC+	2	IN	Z Phase Positive signal	5V
PC-	8	IN	Z Phase Negative signal	

Attention:

1. The output signal of encoder adopt the output way is differential signal 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.

P212: number of spindle teeth

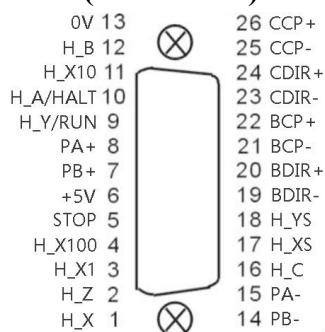
P213, number of encoder teeth

When transmission ratio of spindle and encoder not as 1:1, please modify P212&P213 in Axis parameter when teeth of spindle is not more than teeth of encoder;And set P200-2 in Axis parameter

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.

3.4.3 CN11 MPG/Handhold Box Socket (Male/DB26)



CN11 Handwheel Signal with Male Socket of DB26				
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 input	0V
H_B	12	IN	B-axis selection input	0V
X100	4	IN	*100 rate input	0V
X10	11	IN	*10 rate input	0V
X1	3	IN	*1 rate input	0V
H_A/HALT	10	IN	A-axis selection input/halt stop	0V
H_Z	2	IN	Z-axis selection input	0V
H_Y/RUN	9	IN	Y-axis selection input/run	0V
H_X	1	IN	X-axis selection input	0V
H_C	16	IN	C-axis selection input	0V
H_Xs	17	IN	Xs-axis selection input	0V
H_YS	18	IN	Ys-axis selection input	0V
BDIR-	19	OUT	Negative Direction signal of B-axis	
BDIR+	20	OUT	Positive Direction signal of B-axis	
BCP-	21	OUT	Negative Pulse signal of B-axis	
BCP+	22	OUT	Positive Pulse signal of B-axis	
CDIR-	23	OUT	Negative Direction signal of C-axis	
CDIR+	24	OUT	Positive Direction signal of C-axis	
CCP-	25	OUT	Negative Pulse signal of C-axis	
CCP+	26	OUT	Positive Pulse signal of C-axis	

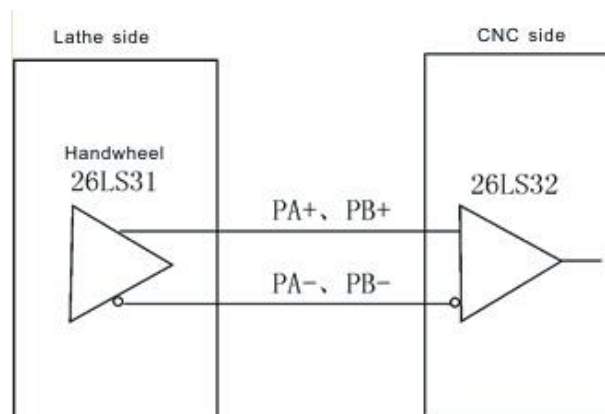
Attention:

1. The input signal for handwheel adopts differential type signal, 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.
4. Voltage of all inner signals is +5V, don't connect voltage, which is over 5V.

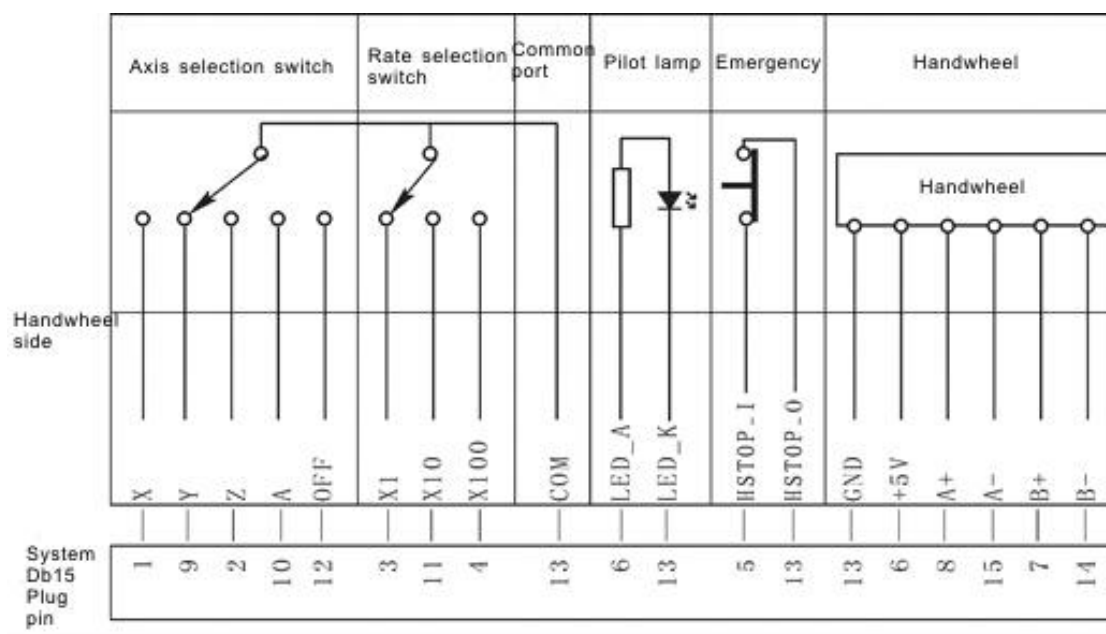
3.4.3.1 Electrical handwheel (Manual pulse generator)

When user connect our handhold box to CN11 Plug, P1 in Other parameter needs to set 1, X Y Z A B C Xs Ys X1 X10 X100 inputs are for axis-selection & rate.

PA+ PB- PA+ PA- are corresponding input signal of handwheel pulse A B.
The input signal of handwheel:



Handwheel contact diagrammatic as:

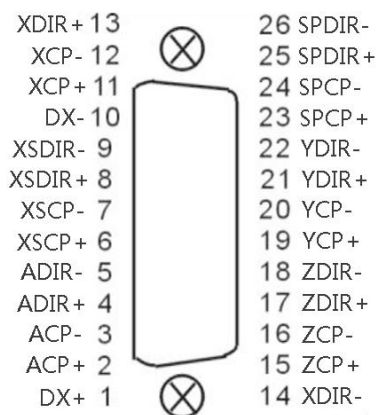


3.4.3.2 Using for External Emergency Stop

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, but it needs to work with B type/E type operation panel.If operational panel(A type/C type) is with handwheel, don't suggest use Handheld box (MPG).

3.4.4 CN21 Pulse Signal Socket (Female/DB26)



CN21 Pulse with Female Socket of DB26			
Signal	pin	I/O	function
DX+	1	-	Positive port of RS485 (RS485+)
DX-	10	-	Negative port of RS485 (RS485-)
ACP+	2	OUT	Positive Pulse signal of A-axis
ACP-	3	OUT	Negative Pulse signal of A-axis
ADIR+	4	OUT	Positive Direction signal of A-axis
ADIR-	5	OUT	Negative Direction signal of A-axis
XSCP+	6	OUT	Positive Pulse signal of Xs-axis
XSCP-	7	OUT	Negative Pulse signal of Xs-axis
XSDIR+	8	OUT	Positive Direction signal of Xs-axis
XSDIR-	9	OUT	Negative Direction signal of Xs-axis
XCP+	11	OUT	Positive Pulse signal of X-axis
XCP-	12	OUT	Negative Pulse signal of X-axis
XDIR+	13	OUT	Positive Direction signal of X-axis
XDIR-	14	OUT	Negative Direction signal of X-axis
ZCP+	15	OUT	Positive Pulse signal of Z-axis
ZCP-	16	OUT	Negative Pulse signal of Z-axis
ZDIR+	17	OUT	Positive Direction signal of Z-axis
ZDIR-	18	OUT	Negative Direction signal of Z-axis
YCP+	19	OUT	Negative Pulse signal of Y-axis
YCP-	20	OUT	Positive Pulse signal of Y-axis
YDIR+	21	OUT	Negative Direction signal of Y-axis
YDIR-	22	OUT	Positive Direction signal of Y-axis
SPCP+	23	OUT	Negative Pulse signal of YS/SP-axis
SPCP-	24	OUT	Positive Pulse signal of YS/SP-axis
SPDIR+	25	OUT	Negative Direction signal of YS/SP-axis
SPDIR-	26	OUT	Positive Direction signal of YS/SP-axis

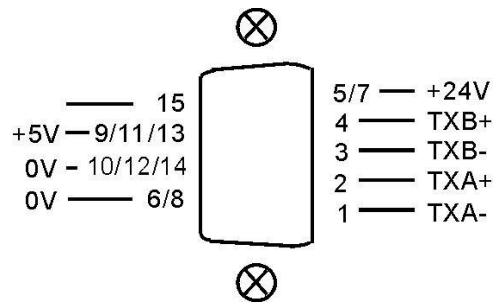
Note:1. The signal cables must adopt shielded twisted pair cable, the length is 20m at most.

2. When use RS485 communication function, 0V_{CNC} must be connected to 0V_{Drivers}.

3.4.5 Debug for SZGH absolute type servo motor

1. Turn on the power supply.
2. Set Mode of XYZABCXSYS axis on Axis parameter (Linear/Rotary Axis).
3. Set Electrical gear ratio for all feeding axes on Axis parameter.
P15=0000000000000001 on Axis parameter, soft limit effectively;
P39=1111111011 on Axis parameter; Returning to zero with floating mode;
4. Parameter set on Other parameter
P300=1111111100; the absolute value function;
P301=92; P302=91; P303=90; P304/P305/P306/P307/P308=131072;
P309/P310/P311/P312/P313 set the coordinate values, such as when the screw pitch with direct of Z axis is 6mm, P311=6000000;
Set P320/P324 when machine with reduction ratio, such as reduction ratio of Z axis is 3:2, thread is 6mm, electronic gear molecule is $5 \times 3 = 15$ (5 without reduction ratio), electronic gear denominator is $3 \times 2 = 6$ (3 without reduction ratio), so:
P322=electronic gear denominator=15, P311=10000000*electronic gear denominator=60000000
5. Set the coordinate axis which corresponds the No.56 parameter in driver parameter
1> P1=1, enter the password;
2> P56=1/2/3/4/5/6/7/8 corresponds to the X/Y/Z/A/B/C/Xs/Ys, P56=3 on Z axis drive;
3> Press "Enter" for a while to save parameter in EP- status;
6. MDI to run the M500 instruction, the small circle in front of the coordinate axis should be changed into "green", means read the absolute encoder data correctly;
7. Set the coordinate value and direction of CNC:
1) Move CNC coordinate manually and remember machine coordinate value;
2) Run M500 instruction on MDI, reading whether the absolute value coordinate is the same as previous coordinate value or not (if change is less than 0.01, also same), modifying the symbol of No.309 No.310 No.311 No.312 No.313 parameter in other parameter, such as the X coordinate is not right, the original is P309=4000000, to modify to P309=-4000000;
3) Repeat 1) 2) test for two times;
8. Setting the zero point of machine: manually move to the position which is nearby the zero point, set the No.314 No.315 No.316 No.317 No.318 parameter in other parameter (Corresponds to X/Y/Z/A/B axis), press "Enter" and press "E" to clear the machine coordinate; and input "EV" to clear multi-revolutions of absolute encoder.
9. Manual move every axis to the limit position/max travel of machine to set the soft limit value in axis parameter;
10. Set the No.41 parameter in other parameter and back up the current system parameter

3.4.6 CN22 Modbus Communication Socket(Female/DB15)



CN21 Pulse with Female Socket of DB26			
Signal	pin	I/O	function
TXA-	1	OUT	A Negative signal of Modbus
TXA+	2	OUT	A Positive signal of Modbus
TXB-	3	OUT	B Negative signal of Modbus
TXB+	4	OUT	B Positive signal of Modbus
+24V	5	OUT	+24V Power
0V	6	OUT	0V Power
+24V	7	OUT	+24V Power
0V	8	OUT	0V Power
+5V	11	OUT	+5V Power
0V	12	OUT	0V Power
+5V	13	OUT	+5V Power
0V	14	OUT	0V Power
DX+	10	-	Positive port of RS485 (RS485+)
DX-	15	-	Negative port of RS485 (RS485-)

3.4.7 IO Ports on IO board

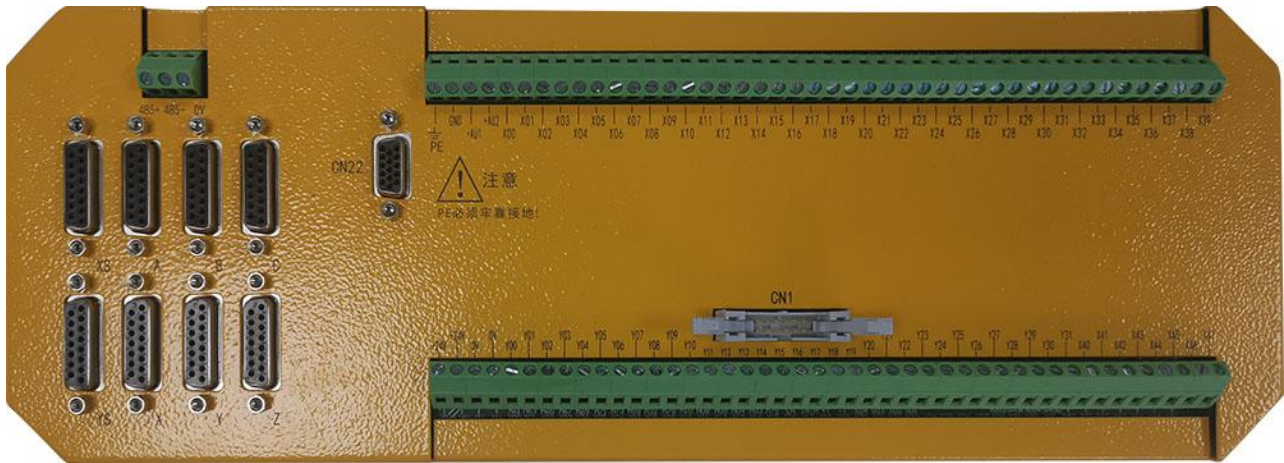
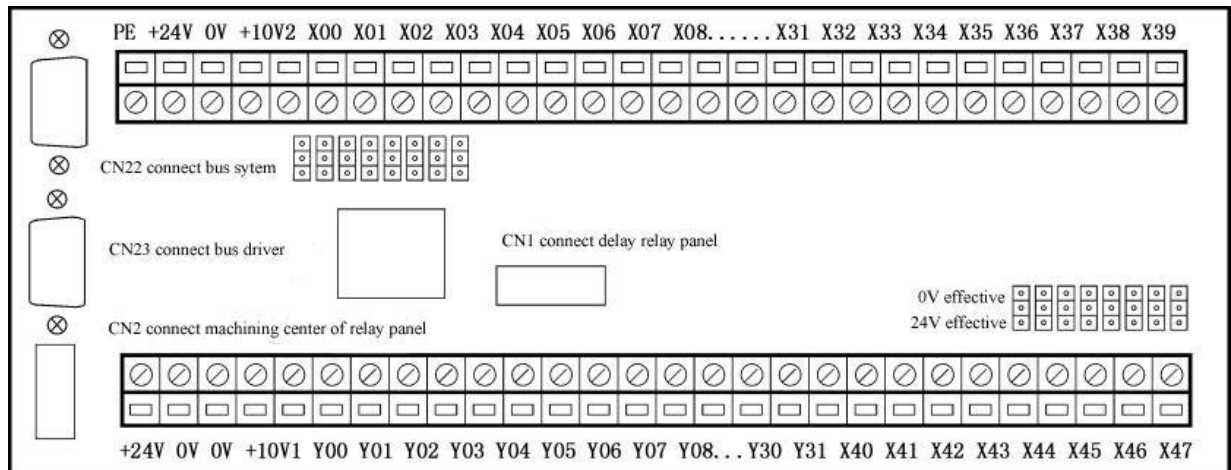


Figure3.3 IO board

3.4.7.1 Inputs & Outputs on IO board



Signal	Address	I/O	function	
PE	-	INPUT	Ground line input/Shield Cover wire input	
GND	-	OUT	GND/0V of Analog Voltage output	
+AV1	-	OUT	1 st Analog voltage output	
+AV2	-	OUT	2 nd Analog voltage output	
X00	T1	IN	User-define input	
X01	T2	IN	User-define input	
X02	T3	IN	User-define input	
X03	T4	IN	User-define input	
X04	T5	IN	User-define input	
X05	T6	IN	User-define input	
X06	T7	IN	User-define input	
X07	T8	IN	User-define input	
X08	A0	IN	Input for A-axis home switch	
X09	-L	OUT	Input for Limit switch in Negative direction	
X10	+L	OUT	Input for Limit switch in Positive direction	
X11	Y0	IN	Input for Y-axis home switch	
X12	X0	IN	Input for X-axis home switch	
X13	Z0	IN	Input for Z-axis home switch	

X14	-	IN	User-define input	
X15	-	IN	User-define input	
X16	YS0	IN	Input for YS-axis home switch	
X17	XS0	IN	Input for XS-axis home switch	
X18	-	IN	User-define input	
X19	-	IN	User-define input	
X20	ALM	OUT	Alarm signal input of 5-8 axes driver	
X21	TOK	OUT	User-define input	
X22	ALM	OUT	Alarm signal input of 1-4 axes drivers	
X23	ALM1	OUT	Alarm signal input of spindle axis	
X24	ALM2	OUT	Alarm signals input of machine tool	
X25	M28/C0	OUT	M28 code input/Input for C-axis home switch	
X26	M24/B0	OUT	M24 code input/Input for B-axis home switch	
X27	M22	OUT	Orientation End	
X28	M18	OUT	M18 code input	
X29	M12	OUT	M12 code input	
X30	M14	OUT	M14 code input	
X31	M16	OUT	M16 code input	
X32	-	OUT	User-define input	
X33	-	OUT	User-define input	
X34	-	OUT	User-define input	
X35	-	OUT	User-define input	
X36	-	IN	User-define input	
X37	-	IN	User-define input	
X38	-	IN	User-define input	
X39	-	OUT	User-define input	
X40	-	OUT	User-define input	
X41	-	OUT	User-define input	
X42	-	OUT	User-define input	
X43	-	OUT	User-define input	
X44	-	OUT	User-define input	
X45	-	OUT	User-define input	
X46	-	OUT	User-define input	
X47	-	IN	User-define input	
0V	-	OUT	0V_24V power_CNC	
+24V	-	OUT	+24V_24V power_CNC	
Y00	M61	OUT	Spindle Orientation	
Y01	M63	OUT	M63 code output	
Y02	M65	OUT	Yellow Indicator(Pause status)	
Y03	M67	OUT	Red Indicator(Alarm status)	
Y04	M69	OUT	Green Indicator(Run status)	
Y05	M71	OUT	M71 code output	
Y06	M73	OUT	M73 code output	
Y07	M59	OUT	Huff/Blower	
Y08	M32	OUT	Lubricate	
Y09	M79	OUT	M79 code output	
Y10	M10	OUT	Chuck	
Y11	M08	OUT	Coolant	

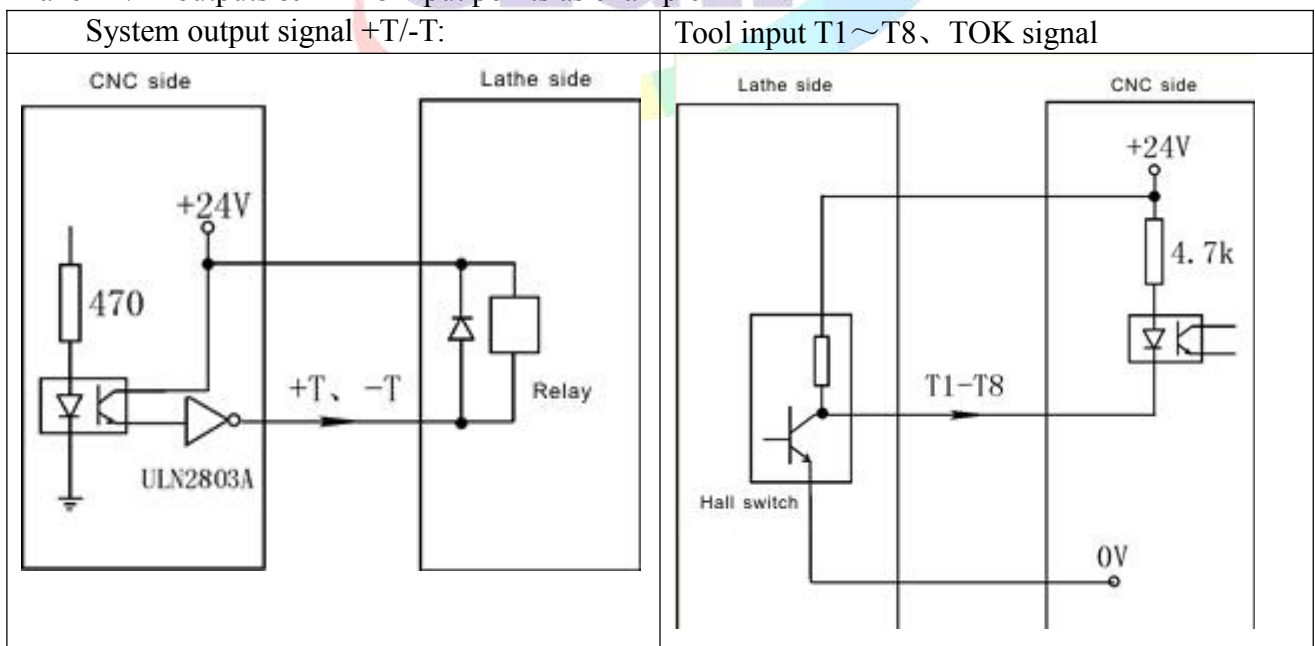
Y12	M05	OUT	Stop of Spindle	
Y13	M04	OUT	CCW rotation of spindle	
Y14	M03	OUT	CW rotation of spindle	
Y15	M75	OUT	Position Mode Shift	
Y16	Ready	OUT	Enable	
Y17	INTH	OUT	Alarm reset	
Y18	+T	OUT	User-defined output	
Y19	-T	OUT	User-defined output	
Y20	S04	OUT	4 th gear output of spindle	
Y21	S03	OUT	3 rd gear output of spindle	
Y22	S02	OUT	2 nd gear output of spindle	
Y23	S01	OUT	1 st gear output of spindle	
Y24	-	OUT	User-defined output/K1 key Control	
Y25	-	OUT	User-defined output/K2 key Control	
Y26	-	OUT	User-defined output/K3 key control	
Y27	-	OUT	User-defined output	
Y28	M203	OUT	CW rotation_2nd spindle	
Y29	M204	OUT	CCW rotation_2nd spindle	
Y30	SPEN	OUT	Spindle Enable	
Y31	SPEN2	OUT	2 nd spindle Enable	

Attention: 1. All inputs & outputs are for system, input is from exterior signal to system, the output is from system signal to external. Valid level of all IOs is 0V(NPN).

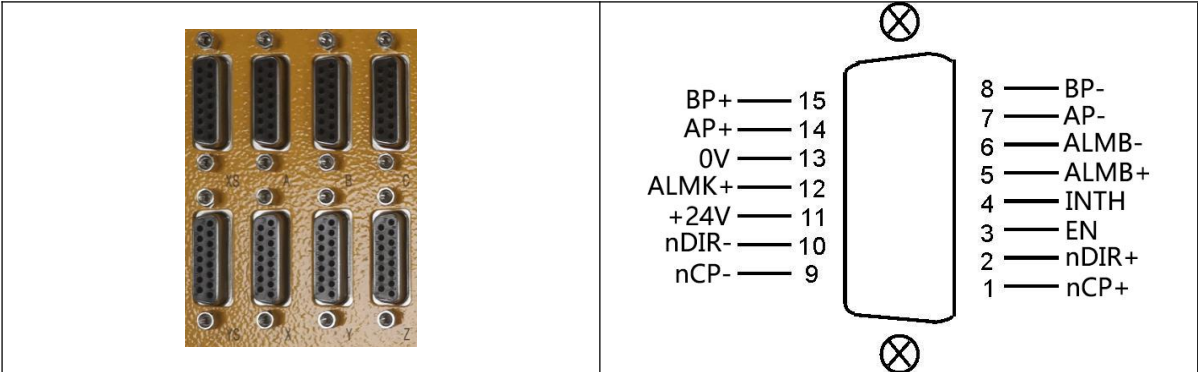
2. Default it must plus a reverse diode in order to cancel reverse current.

3. X00~X07 & X40~X47 can shift NPN/PNP input by inner jumper.

Take +T/-T outputs & T1~T8 input points as example



3.4.7.2 Feeding Axes Pulse output socket(Female/DB15)



Pulse output socket(Female/DB15/X/Y/Z/A/B/C/XS/YS)				
Signal	Pin	I/O	Function	Note
nCP+	1	OUT	Positive Pulse signal	Pulse
nCP-	9	OUT	Negative Pulse signal	
nDIR+	2	OUT	Positive Direction signal	Direction
nDIR-	10	OUT	Negative Direction signal	
EN	3	OUT	Enable	
INTH	4	OUT	Alarm Reset	
ALMB+	5	IN	NC type Alarm signal+	Alarm-NC
ALMB-	6	IN	NC type Alarm signal-	
AP-	7	IN	Negative-A phase Encoder signal	
AP+	14	IN	Positive-A phase Encoder signal	
BP-	8	IN	Negative-B phase Encoder signal	
BP+	15	IN	Positive-B phase Encoder signal	
+24V	11	OUT	+24V power	
ALMK+	12	IN	NO type alarm signal+	Alarm-NO
0V/ALMK-	13	OUT	0V power/NO type alarm signal-	

Note: default alarm wires are for NC type alarm,alarm signals of XYZA axes are connected to X22, alarm signals of BCXSYS axes are connected to X20.If NC,make series connection, if NO type, make parallel connection.



0V	14	OUT	0V Power
485+	10	-	Positive port of RS485 (RS485+)
485-	15	-	Negative port of RS485 (RS485-)

Note: this is for RS485 communication for absolute servo

3.5 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:

3.5.1 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.

3.5.2 Ordinary Problem

3.5.2.1 System can't boot

- 1) check if input power is normal.
- 2) check if power switch is turn on.
- 3) check insurance.

3.5.2.2 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.

3.5.2.3 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.

3.5.2.4 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.

3.5.2.5 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

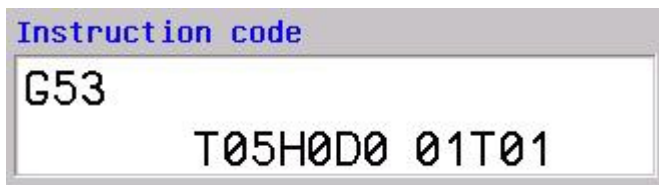
Attention: Because of many kinds of reasons this Manual book may have some mistakes. We will appreciate that you feedback to us or our agents , to provide services and technical support for every customer.

Chapter 4 CNC Machining Center

Note:1.This functions only exist in special CNC controller for machine center.

2.Please confirm type of tool magazine before ex-factory.

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

Press "F7-Set" key on Redeem screen,for set total tools of tool magazine.

4.2 Umbrella Tool Magazine

4.2.1 I/O ports for Umbrella Tool Magazine

Signal	Address	I/O	Function
T01	X00	Input	Detecting if overload of magazine's motor
T02	X01	Input	Detecting if position of Chuck loosing tool
T03	X02	Input	Detecting if level of lubricating oil is low
T04	X03	Input	Detecting if coolant water is lacking
T05	X04	Input	Detecting if position of Chuck tightening tool
T06	X05	Input	Detecting if position of tool magazine forward
T07	X06	Input	Detecting if position of tool magazine backward
T08	X07	Input	Counting of tools
ALM1	X23	Input	Alarm input of Spindle Driver
ALM2	X24	Input	Detecting if overload of cooling motor
M22	X27	Input	Detecting if spindle orientation end
M12	X29	Input	Detecting if safe door of machine tool is open
M14	X30	Input	Detection for pressure alarm of compressed air
M16	X31	Input	External Switch for SP loosen tool
X40	X40	Input	Detecting if overload of motor for removing chips
X46	X46	Input	Activate M01 code(exact stop)
X47	X47	Input	Detecting if overload of fan of spindle
M61	Y00	Output	Spindle orientation
M63	Y01	Output	Rotate CW of tool magazine
M65	Y02	Output	Yellow indicator lamp (Stop)
M67	Y03	Output	Red indicator lamp (Fault)

M69	Y04	Output	Green indicator lamp (Running)
M71	Y05	Output	Rotate CCW of tool magazine
M73	Y06	Output	Forward of tool magazine
M59	Y07	Output	Huff/Blower
M32	Y08	Output	Lubrication
M10	Y10	Output	Spindle chuck loose tool
M08	Y11	Output	Coolant
M05	Y12	Output	Stop spindle/Reset M03&M04
M04	Y13	Output	Rotate CCW of Spindle
M03	Y14	Output	Rotate CW of Spindle
M75	Y15	Output	Shift control mode(Position/Speed) of spindle system
Y24	Y24	Output	Backward of tool magazine
M203	Y28	Output	M203: CW_2nd Spindle
M204	Y29	Output	M204: CCW_2nd Spindle
SPEN	Y30	Output	SPEN: Output Enable of SP servo driver
SPEN2	Y31	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 chapter 4.2.4.

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:

S0 X100% SP000

- 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(Y01),detect input of T08(X07) ;



"TL-CCW":Magazine rotate CCW,Output M71(Y05),detect input of T08(X07) ;

2) M Instruction

M71: Forward of magazine,reset Y24, output M73(Y06);(P23=1 on Tool parameter)

M73: Backward of magazine,reset M73(Y06), output Y24;(P23=1 on Tool parameter)

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(Y00), detect M22 (X27);
- ③ Magazine goes forward:output M73(Y06),detect T06 (X05);
- ④ Chuck loose tool: output M10(Y10), detect T02(X01);

- ⑤ Z axis lifting, height is set by P17 on Tool parameter;
- ⑥ Magazine rotating CW/CCW: output M63/M71, detect T08/X07(count of tools);
- ⑦ Chuck loose tool: output M10/Y10, detect T02/X01(Position of SP-Loose tool);
- ⑧ Z axis falls down to point of tool changing ;
- ⑨ Chuck tighten tool: reset M10/Y10,detect T05/X04(Position of tightening tool);
- ⑩ Tool magazine backward: output Y24, detect T07/X06(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	Address	I/O	Function
T01	X00	Input	Detecting if overload of magazine's motor
T02	X01	Input	Detecting if position of Chuck loosening tool
T03	X02	Input	Detecting if level of lubricating oil is low
T04	X03	Input	Detecting if coolant water is lacking
T05	X04	Input	Detecting if position of Chuck tightening tool
T08	X07	Input	Counting of tools
ALM1	X23	Input	Alarm input of Spindle Driver
ALM2	X24	Input	Detecting if overload of cooling motor
M22	X27	Input	Detecting if spindle orientation end
M12	X29	Input	Detecting if safe door of machine tool is open
M14	X30	Input	Detection for pressure alarm of compressed air
M16	X31	Input	External Switch for SP loosen tool
X40	X40	Input	Detecting if overload of motor for removing chips
X41	X41	Input	Detecting if position of tool holder
X42	X42	Input	Detecting if position of arm(take/exchange/return)
X44	X44	Input	Detecting if position of tool holder lift up
X45	X45	Input	Detecting if position of tool holder fall down
X46	X46	Input	Activate M01 code(exact stop)
X47	X47	Input	Detecting if overload of fan of spindle
M61	Y00	Output	Spindle orientation
M63	Y01	Output	Rotate CW of tool magazine
M65	Y02	Output	Yellow indicator lamp (Stop)

M67	Y03	Output	Red indicator lamp (Fault)
M69	Y04	Output	Green indicator lamp (Running)
M71	Y05	Output	Rotate CCW of tool magazine
M73	Y06	Output	Forward of tool magazine
M59	Y07	Output	Huff/Blower
M32	Y08	Output	Lubrication
M10	Y10	Output	Spindle chuck loose tool
M08	Y11	Output	Coolant
M05	Y12	Output	Stop spindle/Reset M03&M04
M04	Y13	Output	Rotate CCW of Spindle
M03	Y14	Output	Rotate CW of Spindle
M75	Y15	Output	Shift control mode(Position/Speed) of spindle system
Y24	Y24	Output	Arm take/exchange tool, & return to original position
Y26	Y26	Output	Lift tool holder up
Y27	Y27	Output	Fall tool holder down
Y28	Y28	Output	M203: CW_2nd Spindle
Y29	Y29	Output	M204: CCW_2nd Spindle
Y30	Y30	Output	SPEN: Output Enable of SP servo driver
Y31	Y31	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 chapter 4.2.5.

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 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].

P33,CNC don't output Y24 when Z-axis

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:

S0 X100% SP000

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/Y01,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,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,& 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/Y00, detect M22/X27(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(0.001 degree): 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(0.001 degree)

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. If absolute servo system for tool magazine, set P77=367, no need to go home.

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 loosening 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 clasp 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	

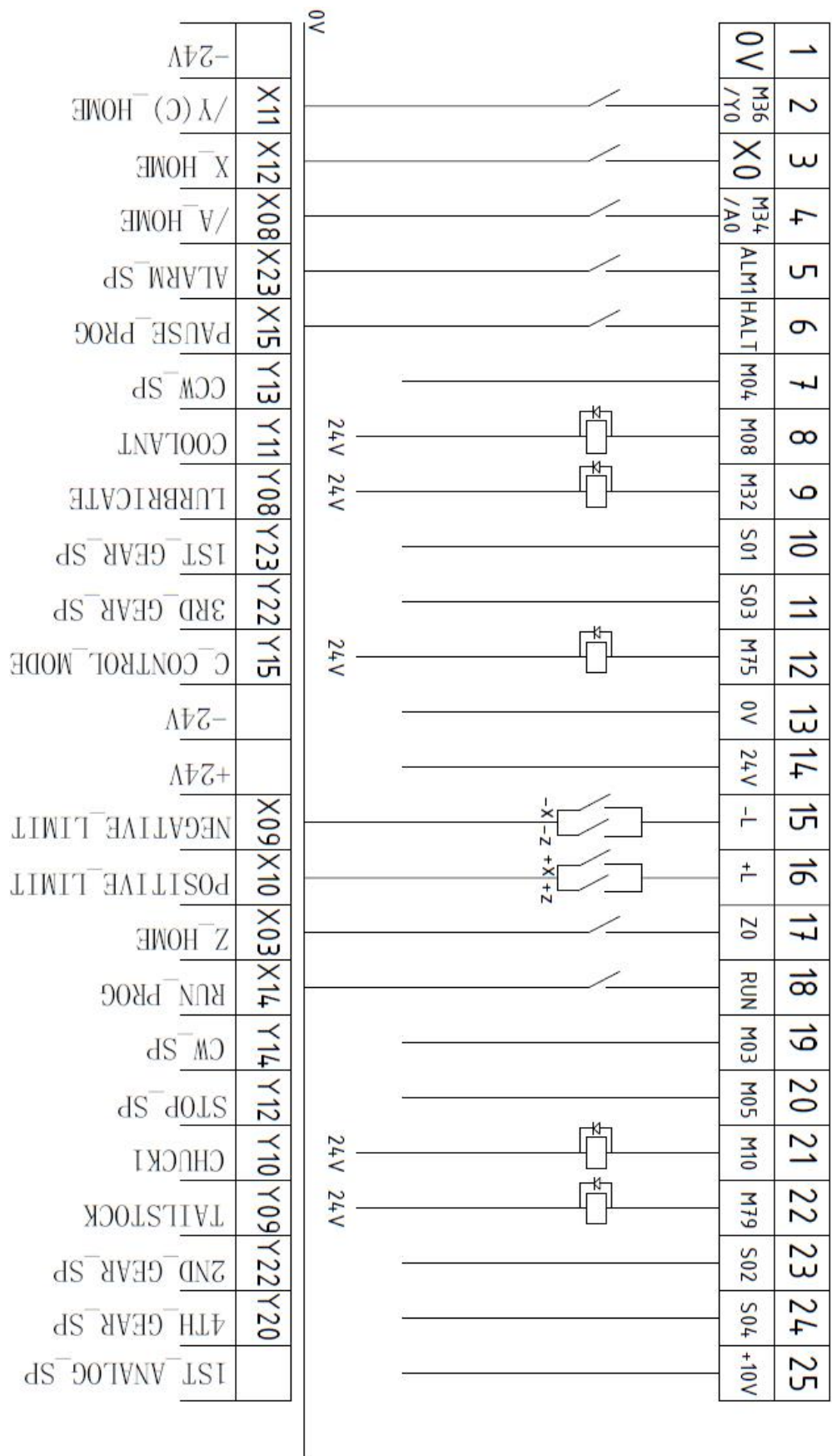
Note:1.Make reversed of feedback signals from tool magazine by IC2803,shown as Fig.4.8.1, also transfer PNP type signals to NPN type signal,which can be acceptable by CNC system.

2.Valid level of all inputs & outputs is 0V.

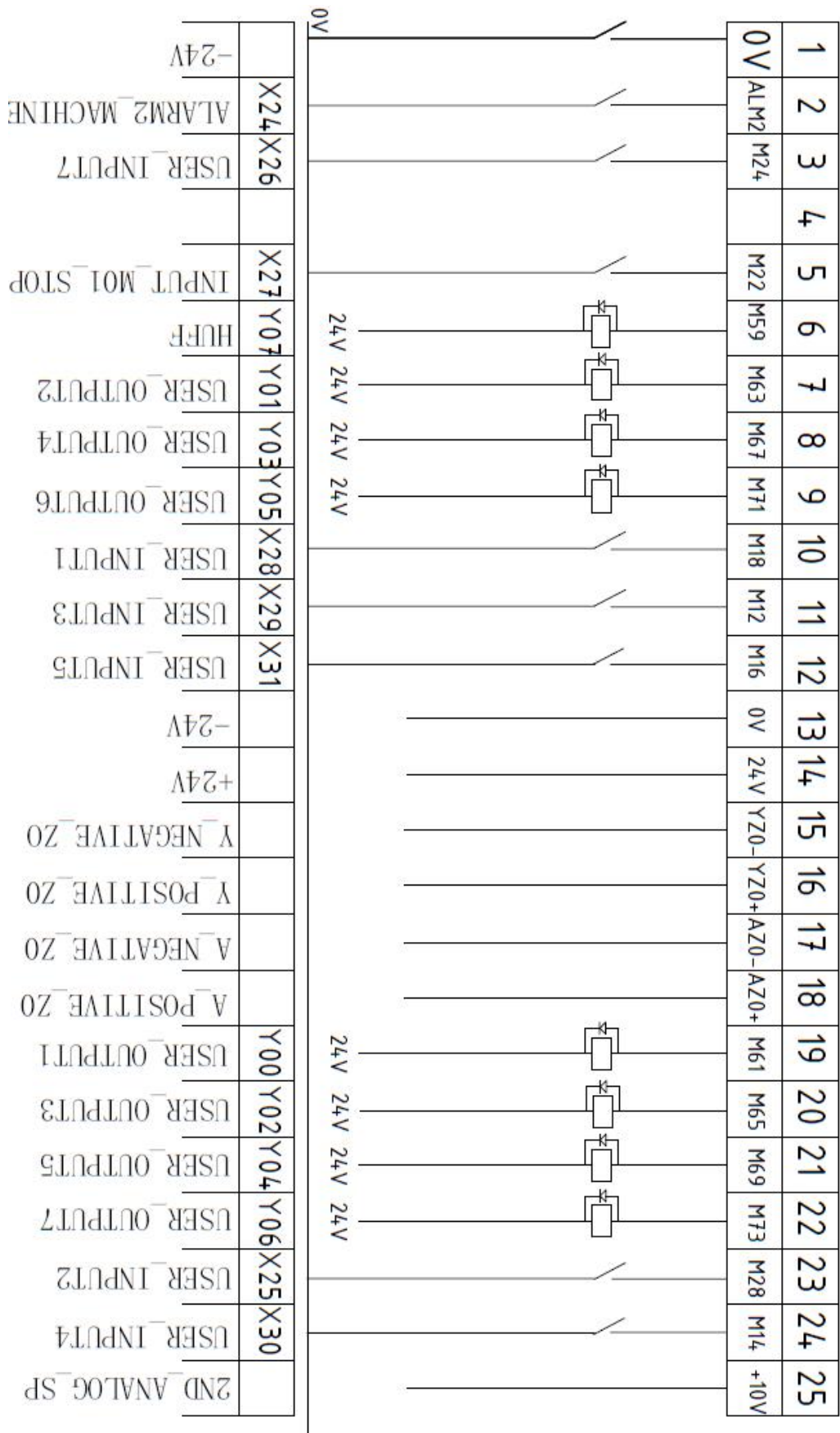


Fig.4.8.1 Transformer board_2803IC

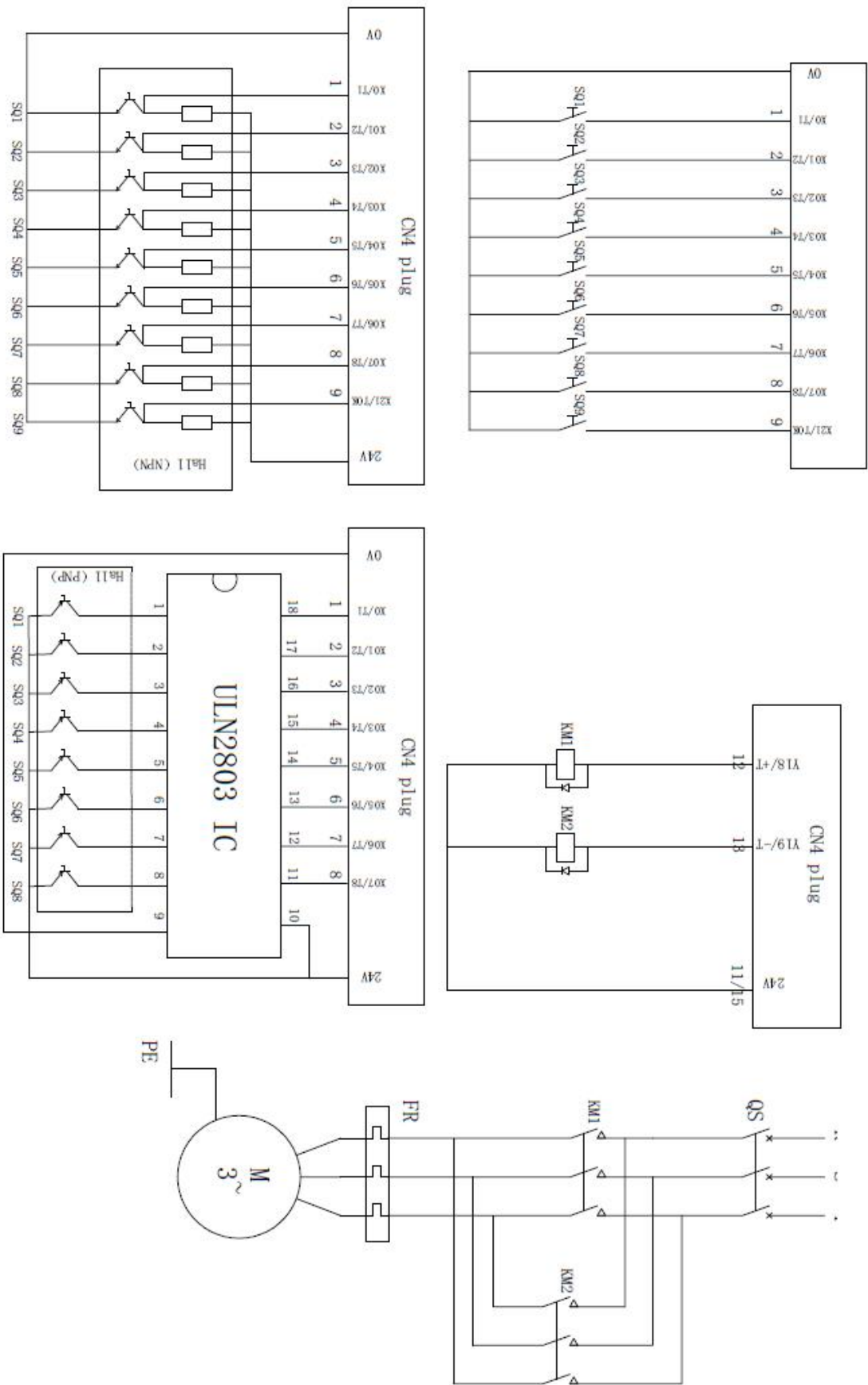
Appendix I: Wiring Diagram of CN3 Plug



Appendix II: Wiring Diagram of CN10 Plug



Appendix III: Wiring Diagram of CN4 Plug



Appendix IV:Wiring Diagram of Bamboo Hat Type Tool Magazine

A. Table of Wiring(Controller To SZGH Spindle Servo Driver)

WIRING DIAGRAM FOR CNC&SPINDLE SERVO				
IO Board_CNC			SP_servo driver(X5I)	
Signal	Add.		PIN	Signal
+24V			1	SEL
0V			10	Ocom
M03	Y14		6	I0
M04	Y13		5	I1
M05	Y12		3	I3
ALM1	X23		9	O0
M75	Y15		2	I4
M61	Y00		4	I2
M22	X27		8	O1
M24	X26		7	O2
PE/Ground line			PE	
			X5M	
+AV1			3	AS+
0V			2	AS-
PE/Ground line			PE	
CN9_CNC			X5P	
PA+	5		4	OA+
PA-	7		7	OA-
PB+	3		8	OB+
PB-	6		9	OB-
PC+	2		2	OZ+
PC-	8		3	OZ-
CN21_CNC			X5P	
SPCP+	23		5	EA+
SPCP-	24		6	EA-
SPDIR+	25		1	EB+
SPDIR-	26		10	EB-

Note: The default resolution of SP_encoder is 1024PPR.

C: Wiring Diagram(SZGH-CNC1000MDi series & Spindle Servo System_V004)

