MACOTECH

User Manual

SZGH-CNC1000TDb(series)
Lathe Control System
-OPERATION-
V2.5

BOOK2

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. GE Fanuc 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 merchantability 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 WARING, CAUTION & 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's memory will be lost. Refer to the maintenance section of the operator's manual for details of the battery replacement procedure.

2. Absolute pulse coder battery replacement

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

NOTE:The absolute pulse coder uses batteries to preserve its absolute position. If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen. When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.

3. Fuse replacement

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

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

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

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

CONTENTS

Chapter 1   Operation

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

1.1 Operational Panel

This system panel is total controller ,which includes 8.4 inches LCD display area , function menu , editing keyboard area & machine control panel.

Fig1.1 SZGH-CNC1000TDb CNC Lathe Controller

1.2 Function Menu

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

1.3 Editing Keyboard

Keys	Name	Description	
RESET	Reset key	CNC reset, stop of the feeding and moving, etc.	
(Address key grid)	Address Key	Address input, Double-address key, switch between addresses	
(Digital/symbol key grid)	Digital Key&Symbol key	Digit input & Symbol Input	
(Cursor arrow keys)	Cursor Move Keys	Move the cursor in different directions.	
PgUp PgDn	Page up/down key	Page up/down on display	*Note: Exchange of coordinate*
Shfit	Shift key	Shift function of key	
Space	Space key	Leave a blank space	
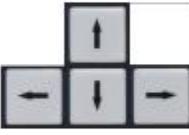
1.4 Machine Control Panel

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

Key	Designation	Explanation	Remarks and operation explanation
Chuck	Chuck switch	Tighten/Loosen Tool of Spindle	Control code: M10/M11 Output Point:PIN21 of CN3 plug
Thumbstall	Tailstock switch	Tailstock Forward/Backward	Control code: M79/M78 Output Point: PIN22 of CN3 plug
Lubricate	Lubricate switch	Lubricate ON/OFF	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, press again, it will shift handwheel rate of *1/*10/*100	
I/O 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 / SP Rate	Spindle override keys	Up/Down rate of SP_Speed	Range is form 5% to 150%,16 gears totally
G00 Rate / G00 Rate	Rate of G00 speed	Up/Down rate of G00_Speed	range is from 5% to 100%,16 gears totally
Feed Rate / Feed Rate	Rate of feeding speed	Up/Down rate of Feeding Speed	range is from 0% to 150%,16 gears totally
SP-CW SP-CCW SP-STOP	Spindle control keys	CW/CCW/Stop of Spindle	Corresponding to M03/M04/M05 codes
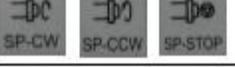
		Rotation of SP in JOG	Point CW/CCW rotation of spindle
		Exchange Tool key	Manual change to next one tool
		Rapid mode	Holding Rapid key+ Manual Feeding Key, for feeding with rapid speed manually. When P38=8 in Other parameter,the key is set to switch of Rapid/Normal.
		Manual Feeding Key	For positive/negative movement of X, Y, Z, A & B axes in MANUAL mode
		Cycle Start Button, for run processing program on Auto, or run MDI	
		Pause Button, for halt processing *Note:only E type operational panel has Pause button.*	
		Emergency stop Driver and motor stop immediately, turns off the spindle, coolant, waits for the rise of button, and initializes values	
		Interface Switch Left side: Normal Mode Middle: F-Halt , stop feeding Right side: FS-Halt, stop feeding & rotation of spindle	
		Handwheel on panel for feeding manually. P1=0 in Other parameter is select handwheel on panel. *Note: only A type & C type operational panel have handwheel(MPG).*	

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

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

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

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

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

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

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

The system adjusts one-level menu operation, intuitive, convenient, shortcut, prompt comprehensive information. System will enter the main screen once power on.

Control Mode:Manual Continous,Manual Increment,Automatic Continous, Automatic Single-Step,Automatic Dry,Handwheel.

Line No. of current processing program

Time
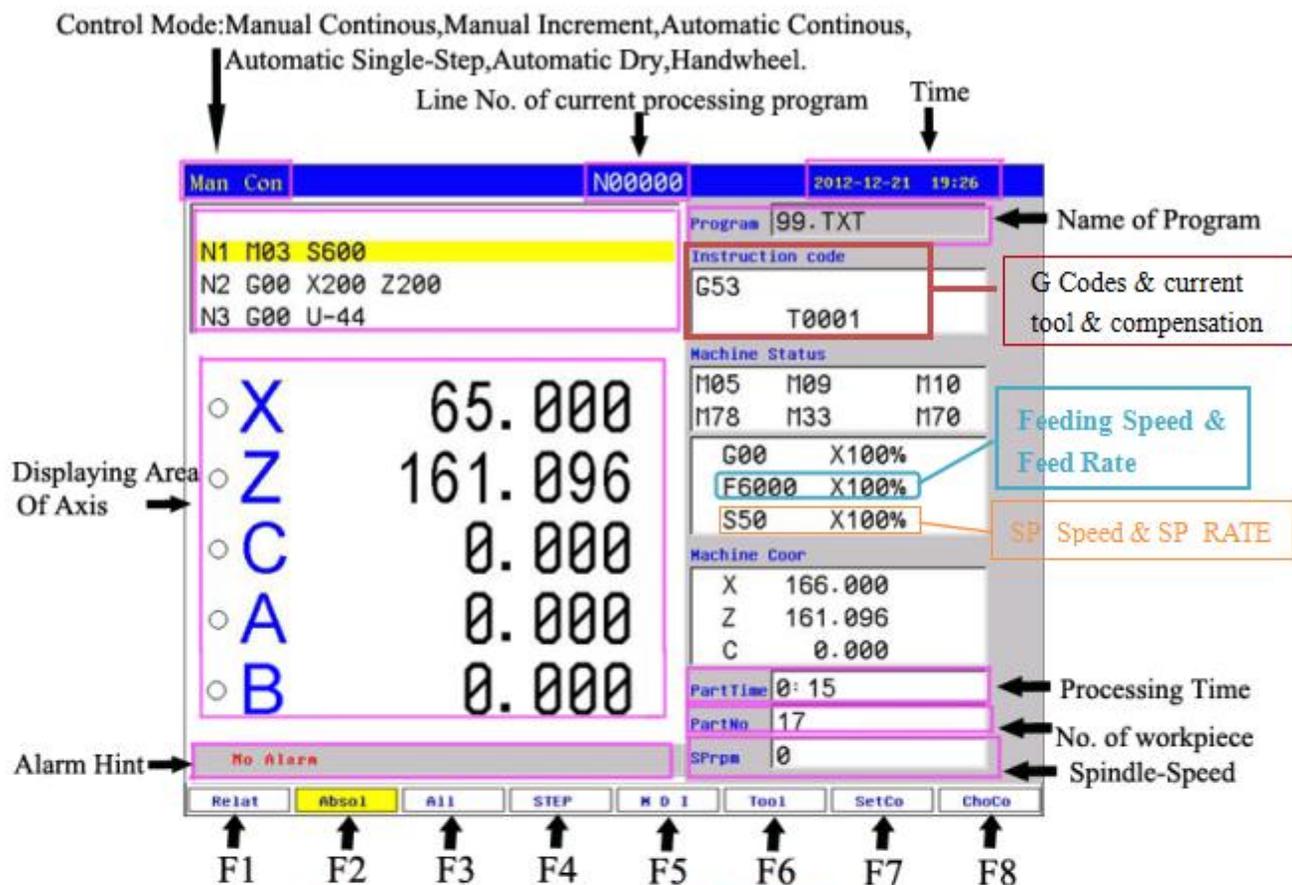
Fig1.2 Displayer

On this main screen, user can check current control mode, curent processing program & current processing block of system. And also can check using G codes & M codes, feeding speed & rate , spindle speed & rate, alarm hint message, workpiece processing time & number, spindle real speed.

User also can select related function rapidly by function keys as following,

Button	Remark	Function
F1	Relat	Enter Relative Coordiante System screen
F2	Absol	Enter Absolute Coordiante System screen(Default)
F3	All	Enter All Coordiante System display screen
F4	STEP	Enter Manual Step mode(Manual Increment/Continous)
F5	MDI	Enter MDI mode (Manual Data Input)
F6	TOOL	Tool Setup
F7	SetCo	Setup workpiece coordiante system(G54-G59)
F8	ChoCo	Choose Workpiece coordinate system(G54-G59)

In manual condition, system could manual process workpiece, cannot run processing program. System adjust coordinate system, tool set, confirm starting point & related loads ON/OFF in Manual condition.

1.5 Manual Operation

1.5.1 Manual Continuous

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

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

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

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

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

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

1.5.2 Manual Increment

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

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

1.5.3 Manual Pulse Generator (Handwheel)

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

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

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

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

Parameters set for handwheel

P1 in Other parameter is set for position of handwheel.

P17 in Speed parameter is set for acceleration / deceleration time constant.

P18 The maximum speed is controlled by No.20(X axis) No.21(Z axis).

Note: 1. SZGH-CNC1000TDb series support handwheel in operational panel & handheld box.

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

Fig1.3 Handheld Box(Manual Pulse Generator)

1.5.4 Manual Reference Position Return

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

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

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

(1) Floating Zero Point Set

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

Parameters set: **In Axis parameter**,

P23: Bit parameter, D3:X ; D4:C(Y) ; D5:Z ; D6:A ; D7:B ; 1 means float zero point;

P24: Distance between reference position & current position in X direction;

P25: Distance between reference position & current position in Z direction;

P119:Distance between reference position & current position in C(Y) direction;

P217:Distance between reference position & current position in A direction;

P318:Distance between reference position & current position in B direction.

Steps of setting floating zero point(home):

1. Enabled function of float zero point of all axes(XZCAB): P23= 11111001;

2. Moving each axes to designated position in order to set floating point easily;

3.Set distance between reference & designated position: P24/P25/P119/P217/P318.

Note: If current position is home of machine coordinate system, no needs to set offset of P24/P25/P119 /P217/P318 for each axis.

(2) Switch For Homing Set

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

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

P23: Bit parameter, D3:X ; D4:C(Y) ; D5:Z ; D6:A ; D7:B; 0 means switch for homing;

P26: Grade of Homing,0: Just hint; 1:No use; 8: Compulsion; 9: Super Compusion;

P27: Mode of Homing, 0 & 2: Detect Z0 signal; 1 & the others, no detect ;

P28: Bit Parameter, Direction & Sequence of Homing;

P29: Bit parameter, Type of switch for homing ;

P30: Range of detecting Z0 signal in X-axis ;

P31: Range of detecting Z0 signal in Z-axis ;

P115: Range of detecting Z0 signal in Y(C)-axis ;

P213: Range of detecting Z0 signal in A-axis ;

P314: Range of detecting Z0 signal in B-axis.

P32: Offset after homing in X-axis

P33: Offset after homing in Z-axis

P116: Offset after homing in Y(C)-axis

P214: Offset after homing in A-axis

P315: Offset after homing in B-axis

In Other parameter: P901: set homing sequence of XYZAB axis.

In Speed Parameter,

P30: X-axis homing speed

P32: Z-axis homing speed

P31: Speed during detecting Z0 signal of X-axis

P33: Speed during detecting Z0 signal of Z-axis

In Axis parameter,

P112: C-axis homing speed

P113: Y-axis homing speed

P211: A-axis homing speed

P312: B-axis homing speed

P114: Speed during detecting Z0 signal of Y-axis

P212: Speed during detecting Z0 signal of A-axis

P313: Speed during detecting Z0 signal of B-axis

Note: Details about parameters set,please check part of Parameters List(see chapter 6.3)

B. Input points for homing in the table

Input Point	PIN	Function
X0	PIN3 of CN3 Plug	X axis homing
Z0	PIN17 of CN3 Plug	Z axis homing
M36/Y0	PIN2 of CN3 Plug	Y(C) axis homing
M34/A0	PIN4 of CN3 Plug	A axis homing
M24/B0	PIN3 of CN3 Plug	B axis homing

Note: 1.when system hasn't Y(C), A axis & B axis, these three input points as function of M36&M34&M24.

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

3. Wiring for homing, please see Apprendix II Wiring Diagram of CN3 Plug.

C. Operation of Return Reference Position

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

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

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

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

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

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

1.5.5 Alignment Tool(Posit Tool)

When machine tool use serval tools to processing , length of each tool is different, so user need to ensure their offset value between tools, also posit tool, which actually is move tool to one position of workpiece, measure coordinate value of this position, and input to CNC system. System will calculates the offset and save to the related offset value of tools to register automatically.

a) Two solutions for posit tool:

A Solution (suggestion)

① Chuck clamp workpiece, set SP_Speed and Feed speed well,run spindle ;

② Select one tool that needs to setup, Example: T0202 ;

③ Turning a cylinder/bore on workpiece in Manual Continous Mode ;

④ Exit Z axis(can't move X-axis), stop spindle ;

⑤ Measure the diameter of workpiece(cylinder/bore);

⑥ Press "Tool" in Function Menu, input "X" & "Enter" , then import the above value of measurement into dialog box, press "Enter" to confirm ;

⑦ Cut endface of workpiece with same steps above, After done, don't move Z-axis ,Stop spindle ;

⑧ Measure distance between endface of workpiece and home of Z-axis(Z0) ;

⑨ Press "Tool" in Function Menu, input "Z" & "Enter" , then import the above value of measurement into dialog box, press "Enter" to confirm ;

⑩ Posit/Setup of No.2 Tool(T02) is already done; Repeat ① – ⑨ to setup /posit/calibrate other tools.

B Solution

1) Chuck clamp workpiece, set SP_Speed and Feed speed well,run spindle ;

2) Select one tool that needs to setup, Example: T0202 ;

3) Turning a cylinder/bore on workpiece in Manual Continous Mode ;

4) Press "Tool" in Function Menu, CNC system will hint a dialog box;

5) Exit both X axis and Z axis , & stop spindle ;

6) Measure the diameter of workpiece (cylinder or bore) ;

7) Press "X" and input above measuring value to dialog box, press "Enter" to confirm ;

8) Cut endface of workpiece with same steps above, stop feeding ;

9) Press "Tool" in Function Menu, CNC system will hint a dialog box;

10) Exit both X axis and Z axis , and stop spindle.

11) Measure distance between endface of workpiece and home of Z-axis(Z0) ;

12) Press "Z" and import the above value of measurement into dialog box, press "Enter" to confirm.

13) Posit/Setup of No.2 Tool(T02) is already done;Repeat (1) — (12) to setup /posit/calibrate other tools.

b) Difference between two methods:

A Solution	B Solution
Must ensure don't exit posit tool axis	Posit tool axis can exit
Tool touch workpiece during exit	Tool don't touch workpiece
Use for posit tool in Z direction	Use for posit tool in X direction

In the above processes, cnc system will save difference value between input measuring value and machine coordinate value to related tool compensation (Redeem) automatically.

If there is error as posit tool, user can adopt the redeem of tool comepnsation that can remove the error.

c) Speical Application

When user needs to use a group of tools to processing serval pieces, that needs using working coordinate system to let tools offset totally.

Method of operation is as follows:

(1) Select anyone tool; Example:T0303

(2) Select corresponding coordinates in MDI: Input G54~G59, press "Start" key;

(3) Turning a cylinder/bore on workpiece in Manual Continous Mode ;

(4) Exit Z axis (cannot move X axis), stop spindle ;

(5) Measure the diameter of workpiece(cylinder or bore).

(6) Press "Setup", input "X" and "Enter", import the measuring value, & "Enter".

(7) Cut endface of workpiece with same steps above ,stop feeding & spindle ;

(8) Measure distance between endface of workpiece and home of Z-axis(Z0) ;

(9) Press "Setup", input "Z" and import the measuring value, & "Enter".

CNC system will save difference between import value and workpiece coordinate value to the corresponding parameters automatically, posit tool in workpiece coordinate also set done well, and the others also set well. Tool setup is okay that user add workpiece coordinate codes (G54~G59) in first line to select the related workpiece coordinate system.

Note: 1.Every tool are independent each other, which has its own coordinate system, so user can can posit any tool at anytime , and also only affect current tool when crashing during processing. Only posit current tool well when crashing or loss of step, all is okay. Method: In the status of G53, operate method of operation at c) speical application.

2. Before posit tool,please ensure that CNC system can ATC normllay.

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 manully 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 2. 4)

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

Relative Coordinate	Absolute Coordiante	All Coordiante			
•U 0.000	•X 0.000	Relative U 0.000 W 0.000		Machine X 0.000 Z 0.000	
•W 0.000	•Z 0.000	Absolute X 0.000 Z 0.000		Distance to go X 0.000 Z 0.000	

1.6.1 Automatical Processing Mode

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

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

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

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

Table of Status Display about Manual/Auto

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

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

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

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

There are two kinds of figure, also two types of lathe machine, one is Front Turret lathe machine, the other is rear turret lathe machine, P3 in Tool parameter is set for type of lathe machine.

Front Toolpost Coodriante System	Rear Toolpost Coodriante System
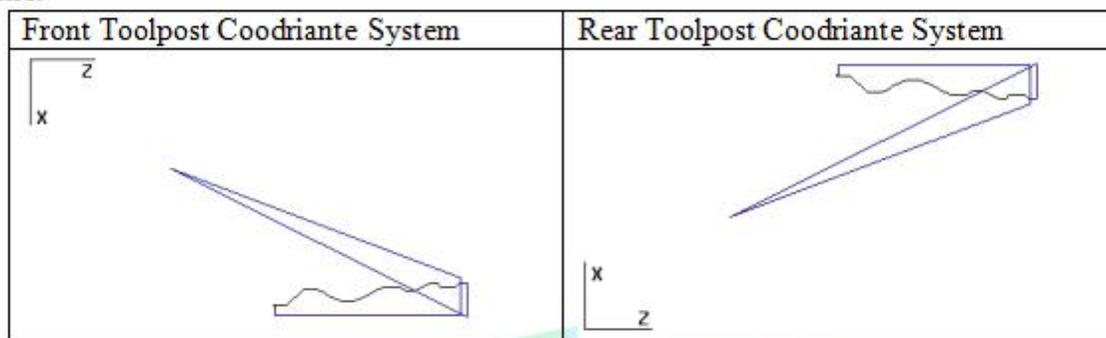
1.6.2 Processing at arbitrary program line or with arbitrary tool

1.6.2.1 Start from "nth" line(block)

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

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

2. Firstly of all, system will move the starting point of "nth" block with speed which is set by P5 in Speed parameter, then run the program normally.

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

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

1.6.2.2 Start from "N**" line

The system support that program can start from "N**" line(N** is 4 digit marked line) . At automatical process condition, press "N" to popup 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 P5 in Speed parameter, then run the program normally.

1.6.2.3 Start from "nth" tool

The system has a function to start program from someone tool("nth" tool). At automatical process condition, press "G" and input number of tool , press "Enter" to confirm. Press "Run" to process program at the tool you import.

Note: *1.only input number of tool, not need to input number of compensation; Eg.: T0304, just import "03" ;*

2. Firstly of all, system will move the starting point of "nth" block with speed which is set by P5 in Speed parameter, then run the program normally.

1.6.3 Start Program

Start program must in the mode of "Auto" , press "Auto" key to enter mode of Auto, there are two methods to start program, as follows,
   (1) Press "Start" key in the operational panel.
   (2) Fix external switch to Run port(PIN18_CN3/PIN8_CN6/PIN9_CN11)
  Note: PIN9_CN11 can be used as Run port, P33 in Other parameter is set for this function.

1.6.4 Halt Program

There are five methods to stop/pause program , as follows,
   (1) Instructions of program: M00, M01, M02, M30, M20.
   (2) Press "Single" to run a current block and stop.
   (3) Interface switch turn to the middle or right side.
   (4) Press "Reset" to stop all the actions of program.
   (5) Fix external switch to Halt port(PIN6_CN3/PIN6_CN6/PIN10_CN11).
Note: PIN10_CN11 can be used as Halt port, P34 in Other parameter is set for this function.

1.6.5 Emergency Stop

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

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

After "Emergency Stop" during processing, which will affect difference between machine tool coordinate system & actual position of machine tool, in order to ensure coordinate system is same to actual position of machine tool, it is best to do manual return reference position(homing) before processing.

1.6.6 Alarm

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

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

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

There are also some user-defined alarm as following

Alarm Hinting Message	Auxilary Relay in PLC
No.0 Alarm	M80
No.1 Alarm	M81
No.2 Alarm	M82
No.3 Alarm	M83

No.4 Alarm	M84
Protect Door Is Open	M85
No.6 Alarm	M86
No.7 Alarm	M87
Loss of Lubricate Oil	M88
No.9 Alarm	M89
No.10 Alarm	M90
No.11 Alarm	M91
+5V Undervoltage	M92
+24V Undervoltage	M93
No.14 Alarm	M94
No.15 Alarm	M95

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

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

In Other parameter:

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

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

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

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

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

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

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

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

1.6.7 Indicator Light Output

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

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

1.6.8 DNC function

Storage room of SZGH cnc system is 128Mbit, user can adopt RS232-DNC or USB-DNC function to run the processing program that is greater than the remainder storage. RS232 port & USB port are in the front/rear of CNC990TDb series controller.

1.6.8.1 RS232-DNC

1. Connect PC and CNC system well with the dedicated communication line , & set communication rate by P37 in Other parameter;

2. Use the dedicated communication software(SZGHCNCCS) on PC to set the related communication port and rate. Press "Transmit CNC", select the program file to process linked, enter the status of sending program file.

3. To enter the interface of program file on CNC system, press "L" to enter the status of linked process, and program will display "RS232--DNC", press "Start" to running carry out linked process in the automatical status.

4. Turn "Interface switch" to middle or right to stop the running system in the process of linked

process, press "E-Stop" or "Reset" to exit link of DNC.

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

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

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

1.6.8.2 USB-DNC

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

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

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

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

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

1.7 External Electrical Connection

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

1.7.1 Limitation

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

1.7.1.1 Software limitation

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

In Axis Parameter

P11: bit parameter, software-limitation of each axis is set alone.

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

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

P3: Max Range in X-Negative Direction

P4: Max Range in X-Positive Direction

P5: Max Range in Z-Negative Direction

P6: Max Range in Z-Positive Direction

P117: Max Range in C(Y)-Negative Direction

P118: Max Range in C(Y)-Positive Direction

P215: Max Range in A-Negative Direction

P216: Max Range in A-Positive Direction

P316: Max Range in B-Negative Direction

P317: Max Range in B-Positive Direction

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

1.7.1.2 External Switch for limitation

Input Point of Limitation

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

Type of Limitation Switch

In Axis parameter:

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

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

Wiring of Limitation
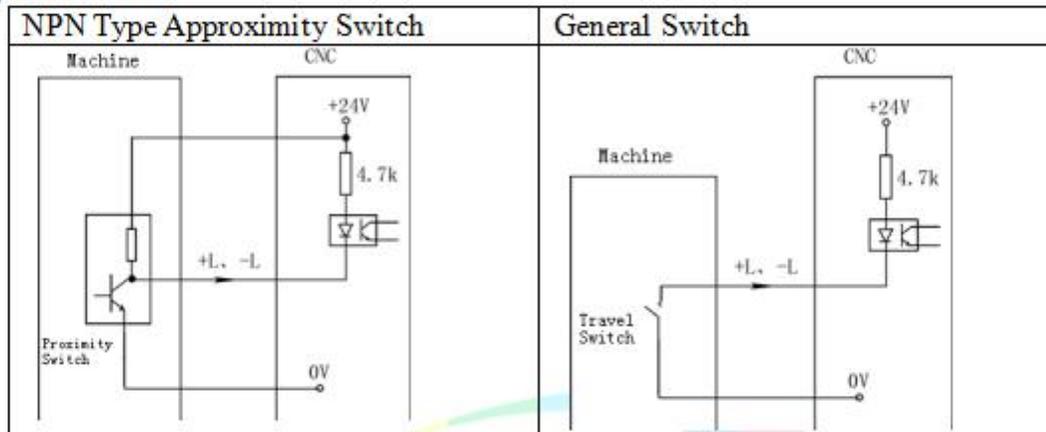
1.7.1.3 Suggestion Usage

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

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

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

2.1 Limitation in negative or positive direction of X axis use one input port , -L ;

2.2 Limitation in negative or positive direction of Z axis use one input port , +L ;

In Axis parameter:

P21, Type of Switch in X direction [0: NO type, 1: NC Type]

P22, Type of Switch in Z direction [0: NO type, 1: NC Type]

1.7.1.4 External Switch for Power ON/OFF

It needs to use one contactor(KM1) & two switches for turn on & turn off power, one is NO type, which is for turn on power, another is NC type, which is for turn off power.
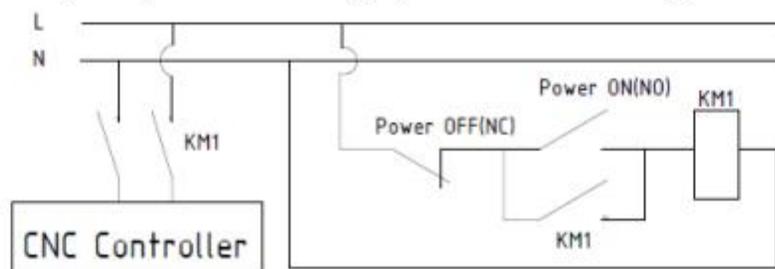
Fig1.4 Wiring Diagram for Power ON/OFF

1.8 Diagnosis

Press "Diagnosis" key or press "Parameter"key twice one the panel to enter the diagnosis interface.

Press "F2" & "PgDn/PgUp"or " ↑↓ " to check the status of inputs and outputs.
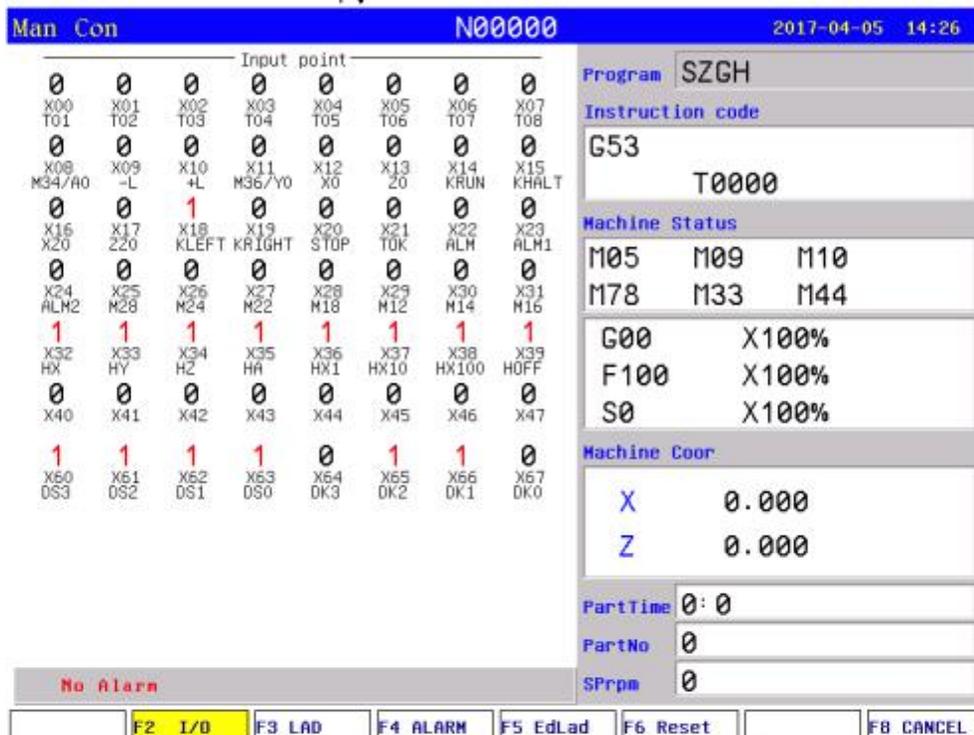
Fig1.8.1 System Diagnosis Interface(Input signal)
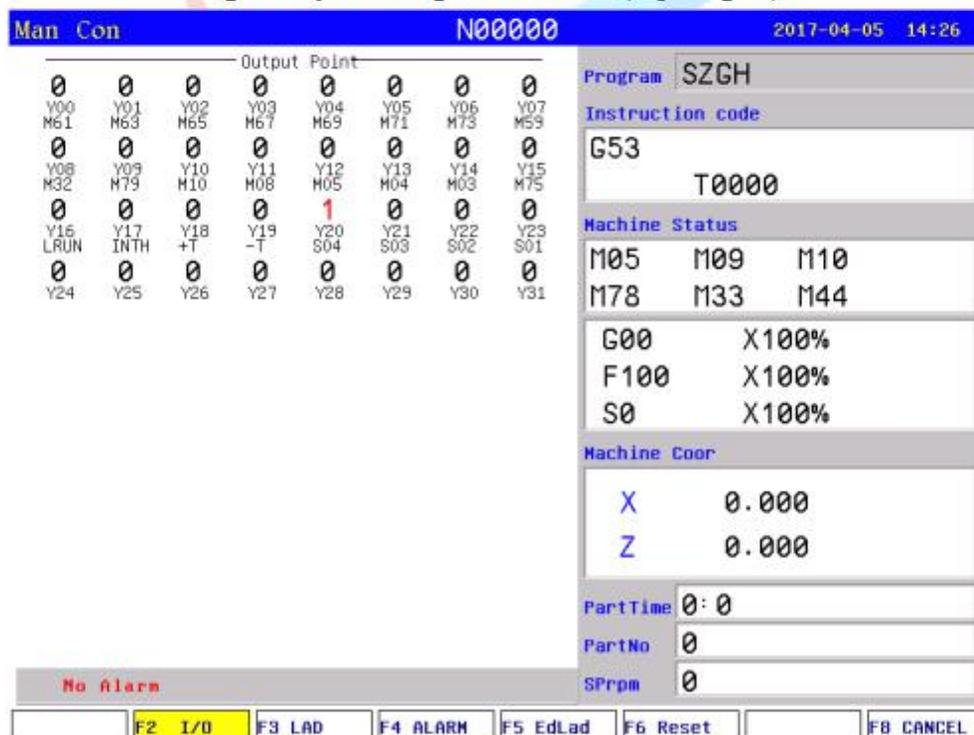
Fig1.8.2 System Diagnosis Interface(output signal)

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

Press "F3" key diagnosis screen to enter interface of check condition of PLC.
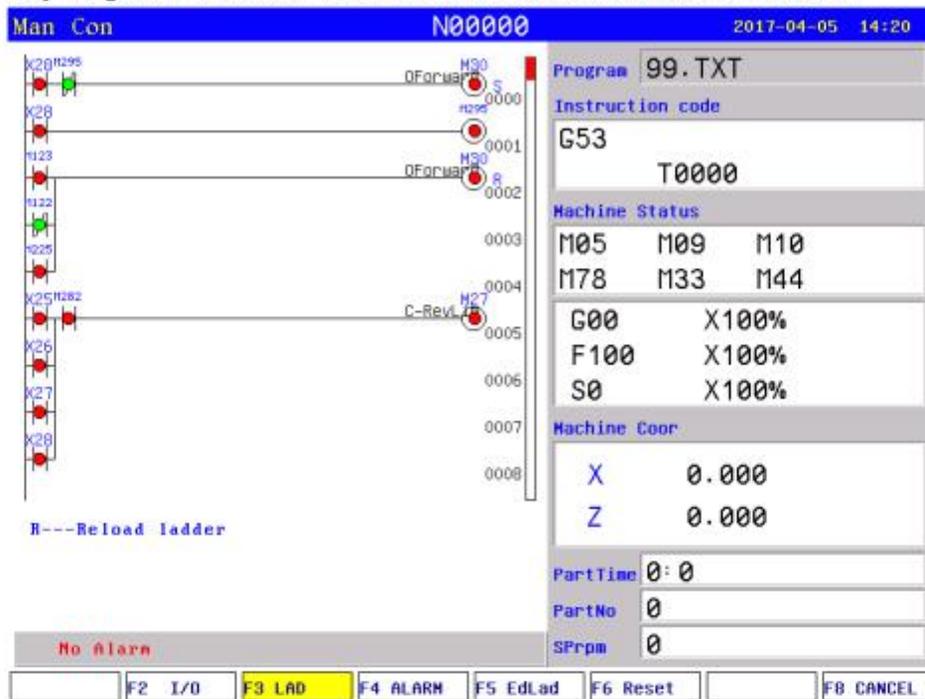
Fig1.8.3 Conditon of Inner Register & IOs

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

"Green" means the register is valid, "Red" menas the register is invalid.

Press "F5" key on diagnosis screen to enter interface of edit ladder of PLC.
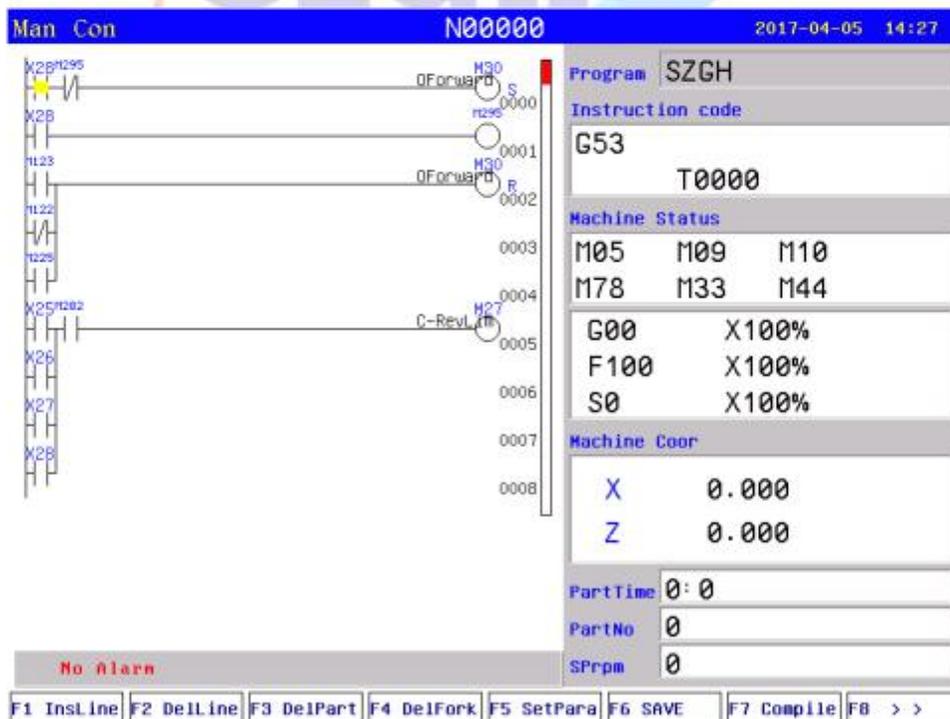
Fig1.8.4 Editing Screen of Inner Ladder

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

Press "R" key on condition screen of PLC, PLC will work immeditately & no needs to reboot.
Note: when P1 in Password parameter set to Disable, and then user can check & edit inner ladder.
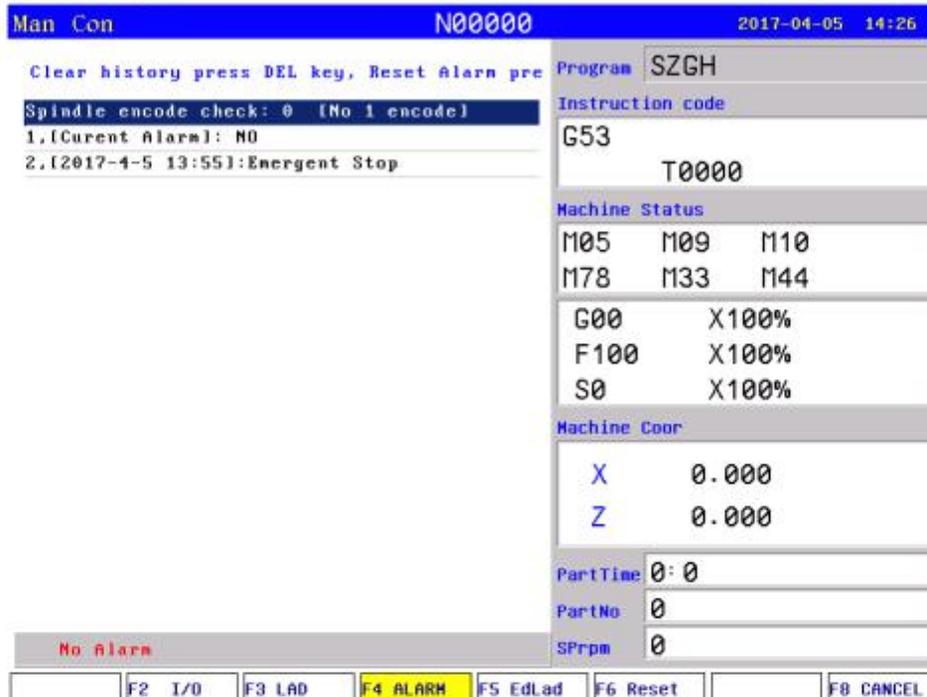
Fig1.8.5 System Diagnosis Interface(Alarm messages)

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

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

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

1.9 Programming Operation

Press "Program" to enter into status of Programm as following.
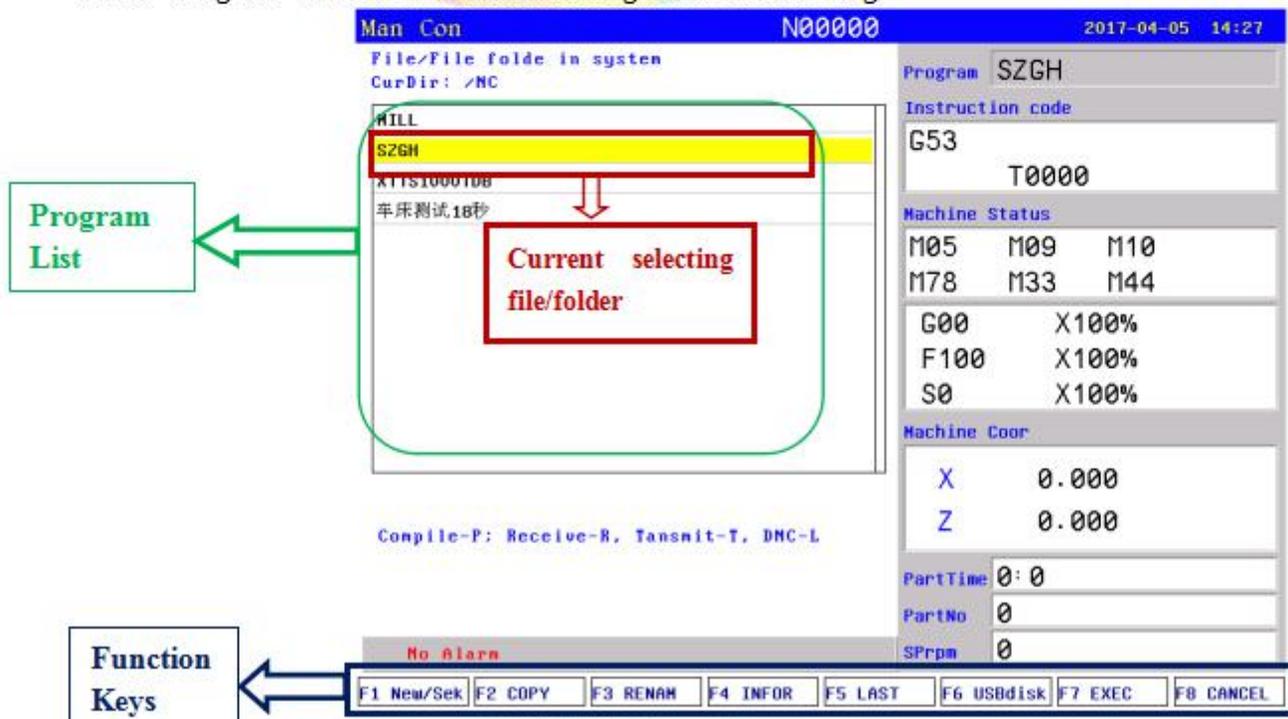
Fig1.9.1 Interface of Program

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

At program list,press "PgDn/PgUp"or ⌊↑, " to select program/file.and then press "Enter" to enter current program.

Name	Function
New/Sek	Press "F1" key to New/Search a program
COPY	Press "F2"key to Copy selecting program(System ↔ USBdisk)
RENAM	Press "F3" key to Rename current selecting program
INFOR	Press "F4" key to hints size of program& remain space of system
LAST	Press "F5" key to return to last level
USBdisk	Press "F6" key to open USB-disk
EXEC	Press "F7" key to execute current program
CANCEL	Press "F8" key to cancel or return
Compile-P	Press "P" key to compile current program
Receive-R	Press "Rkey" 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 "N"key and popup a dialog box to input the name of program, if the name is existing, the existing program is called up; If the name isn't existing, the system will build a new program.

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

Build a new program or select a program and press "Enter" to entering the editing interface.Press "F7" button to shift to function interface.
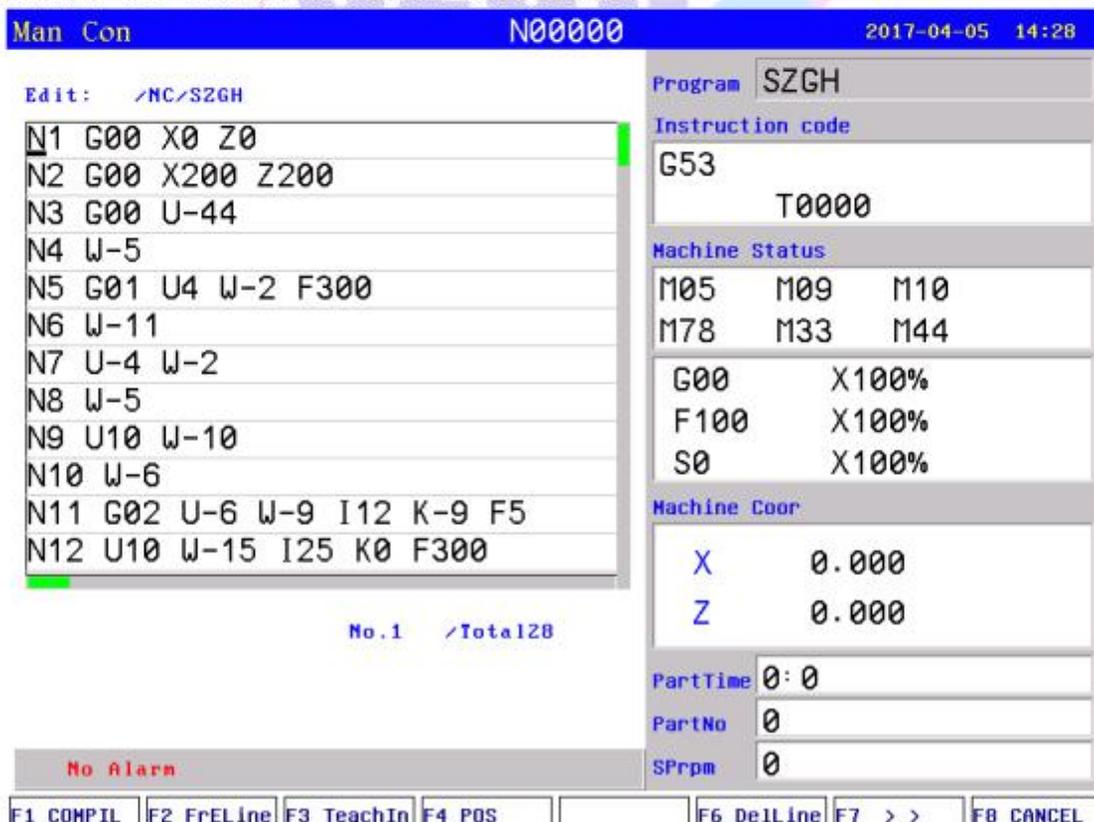
Fig1.9.2 (1)Editing interface of Program
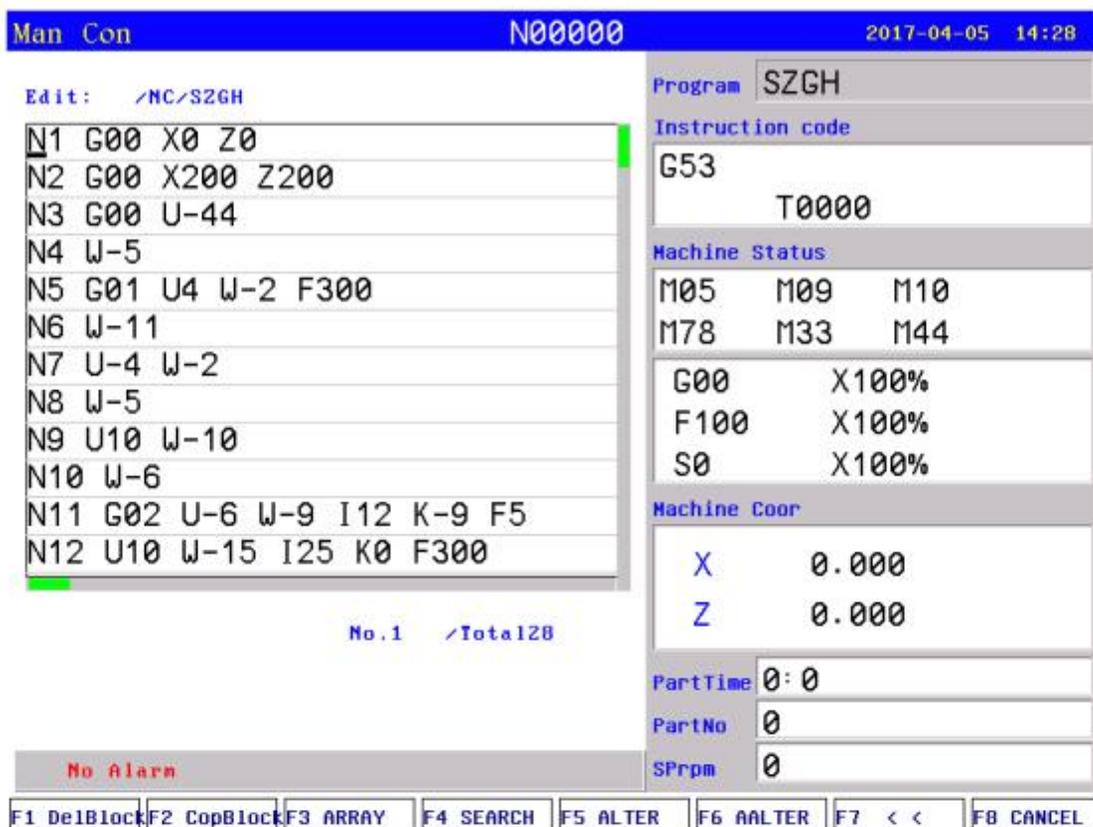
Fig1.9.3 (2)Editing interface of Program

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

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

(1) Location of current cursor:

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

Press "Pgup" to the last page.

Press "Pgdn" to the next page.

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

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

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

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

(6) "Rapid" key Overlay operation:

First Function Keys	
Superposition	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	Delect specified blocks from current line to input line
F2	Copy specified blocks from input begin line to input last line
F3	Array all bloacks 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 popup a dialog box to import a new name of program, to copy which is the same content but different name , in order to modify, rename and spare.

1.9.3 Delete

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

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

1.9.4 Rename

Press "↑ ↓ " in main interface of Program, to select program which need to rename and press "F3" to popup a dialog box to import a new name.

1.9.5 Information

Press "↑ ↓ " in main interface of Program, to select program which need to check and press "4" to popup a dialog box to check the size of current program and remain space of the system.

1.9.6 Compile

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

Note: Before run program, it should compile successfully.

1.9.7 Folder management

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

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

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

Press "A" go to the last folder.

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

1.9.8 Excute Program

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

1.9.9 Communication

The system could deliver files with RS232 serial protocol .

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

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

Delivery (Transmit)

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

Reception

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

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

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

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

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

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

1.9.10 U-disk management

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

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

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

We prepare one special USB-disk for user, which includes user manual & related software.

Fig1.9.4 USB-Disk

Press "F6"-USBdisk to enter the U-disk in main interface after U-disk is connected to USB port in the front of panel. Press "F6"-SYSTM again to back interface of system.
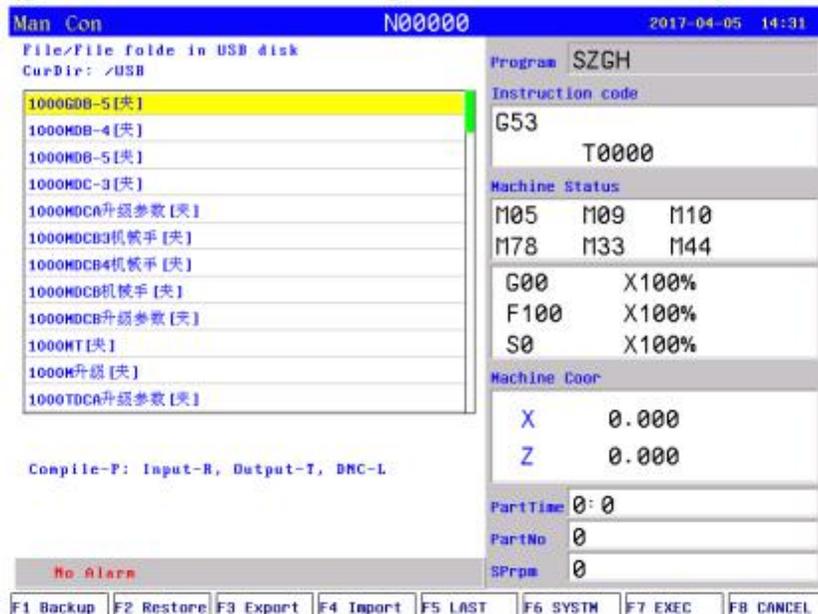
Fig1.9.4 Mian Interface of USB-disk

1.9.10.1 Function Keys of USB-disk

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

1.9.10.2 Management of Processing Program

Copy the files or folder of U-disk into system

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

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

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

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

1.9.10.3 Management of Parameters & Software

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

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

Note: Prepare a special empty U-disk for manage parameter files & PLC files of system better as Parameter files is lots of about several dozens, Or setup a folder in U-disk on your computer firstly, open the folder of U-disk on system before backup parameter files & PLC files into the folder of U-disk.
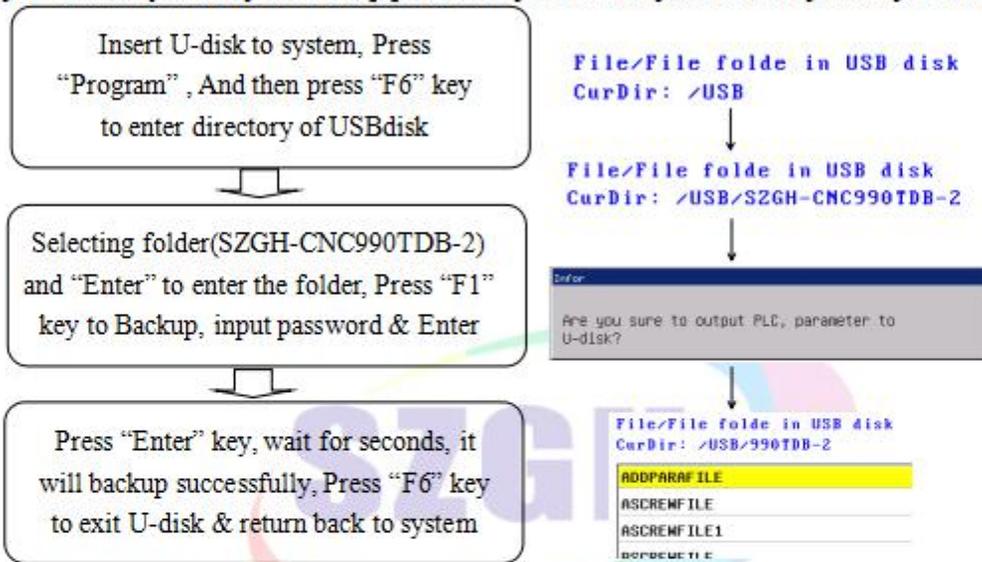
Fig1.9.5 Steps of Backup PLC & Parameter to U-disk

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

Note: Please put parameters & PLC for upgrade to a folder, which is better to avoid restore wrong files in U-disk into CNC system, and result to damage inner files & system.
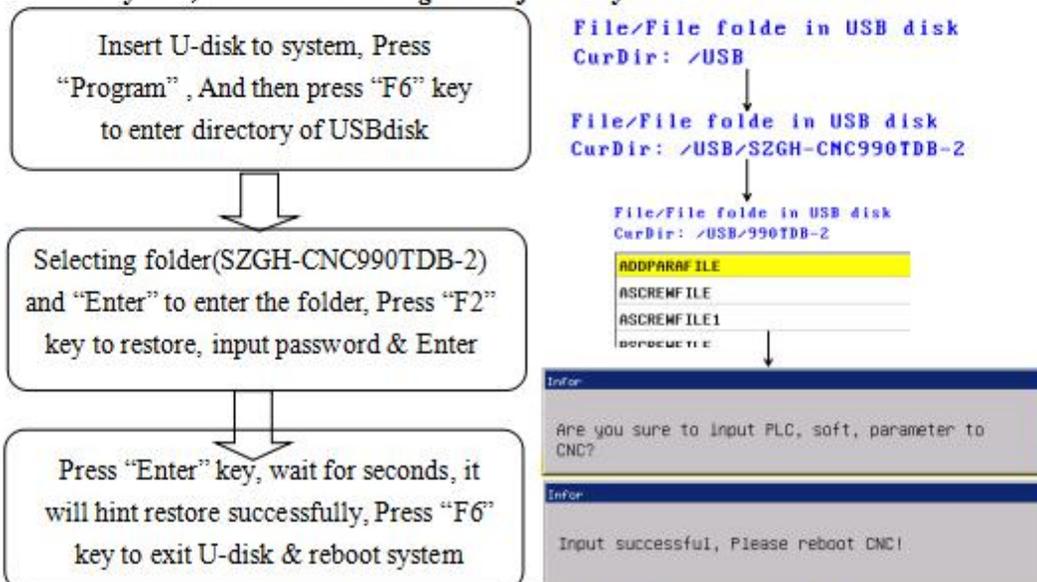
Fig1.9.6 Steps of Restore PLC & Parameter to U-disk

Chapter 2   Parameter List

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

Man Con	N00000	2017-04-06 17:19

Program **SZGH**

Instruction code

G53
        T0000

1,(^d)Depth of	**50**
2,(e)Escaping A	10
3,G71/G72/G73 d	0
4,(^u)X_Distanc	20
5,(^w)Z_Distanc	20
6,(d)The number	1
7,(^i)X_Distanc	10
8,(^k)Z_Distanc	10
9,(^k)Deptch of	50
10,(e)Return Am	10
11,(m)Repetitiv	1
12,(r)Chamferin	10
13,(a)Angle of	60

Machine Status

M05	M09	M10
M78	M33	M44

G00	X100%
F100	X100%
S0	X100%

Machine Coor

X	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 interface & select kind of parameter list, press " Down arrow ", "Up arrow", "PgDn", "PgUp" to select one parameter that need to alter, Press "Enter" enter key , popup dialog box,after alter well, press "Enter" for ensure parameter is set well.

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

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

2.1 User Parameter

P	Parameter	Ex-Value
1	(^d)Depth of Cutting in G71/G72[Radius Designation] (10um)	50
2	(e)Escaping Amount in G71/G72[Radius Designation](10um)	10
3	G71/G72/G73 do Finishing Automatically [1:Yes, 0:No]	0
4	(^u)X_Distance&Direction of finishing allowance of G71/G72/G73 (10um)	20
5	(^w)Z_Distance&Ddirection of finishing allowance of G71/G72/G73 (10um)	20
5-1	Auto_Feeding/Escaping Direction of G71/G72/G73[0:No, 1:Yes]	
6	(d)The number of division in G73	1
7	(^i)X_Distance and direction of relief of G73[Radius Designation] (10um)	10
8	(^k)Z_Distance and direction of relief of G73(10um)	10
9	(^k)Deptch of Cut in Z direction of G74/G75(10um) [Without Sign]	50
10	(e)Return Amount of G74/G75(10um) [Radius Designation]	10
11	(m)Repetitive Count in Finishing of G76	1
12	(r)Chamfering amount of G76(1/10 lead)	10
13	(a)Angle of tool tip in G76(degree) [0~180°]	60
14	(^dmin)Minimum Cutting Depth of G76(10um) [Radius Designation]	20
15	(d)Finishing allowance of G76(10um)	10
16	Programming mode of X axis [1: Radius,0: Diameter]	0
17	Interlock between Running Program & SP-Rotating [1:Yes, 0:No]	0
17-1	Interlock between Running Program & SP_Chuck[1:Yes, 0:No]	0
18	Times of auto-cycle running(M20)[<0: Endless Loop]	-1
20	Length of chamfering in G92(1/10 lead)	0
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
24	Q(△d)infeed times in G76 [8:Times of rough cutting]	1
33	Type of Detect SP_Speed Reached(0:M69 Realy, 8:SP_Encoder)	1
34	Allow error of SP_Speed that detected by SP_Encoder(RPM)	1
200	Waiting time to screensaver [>=2minutes]	1
201	Delay time before detect zero pulse when threading(ms)[>100]	1
203	Inspect overcut when tool radius offset (2131970:Yes, 34818:No, 6326274: Pre-read)	23103490
210	Type of Graphic display area(8:manual,other:Automatic)	0
211	Display X-axis Negative area (1:Yes, 0: No)	1
212	Display X-axis Positive area (1:Yes, 0: No)	1
213	Display Y-axis Negative area (1:Yes, 0: No)	1
214	Display Y-axis Positive area (1:Yes, 0: No)	1
215	Display Z-axis Negative area (1:Yes, 0: No)	1
216	Display Z-axis Positive area (1:Yes, 0: No)	1
230	Running program through input point (+4+8+16+32+64+128: X26-X31)	0
231	Mode of "Delete" key [0:backward deletion,1:Forward delete]	0
232	Detect SP_Zero position before tapping [18:Yes,0:No]	18
233	G06 Circle teaching function[0:No, 1:Yes]	0
234	Activate Program Back Function with Handwheel[+8:Yes, Other: No]	9

Explanation about User Parameter(processing parameter)

1,(^d)Depth of Cutting in G71/G72[Radius Designation](10um)

It is set for every default infeed in X-axis direction when W(Δd) isn't sepcified in G71 code, radius designation; unit:0.01mm.

It is also set for every default infeed in Z-axis direction when it isn't set in G72 code; unit:0.01mm

2,(e)Escaping Amount in G71/G72[Radius Designation](10um)

It is set for every default backward (radius) in X-axis direction when it isn't be set in G71 code;unit:0.01mm.

It is set for every default backward in Z-axis direction when it isn't be set in G72 code;unit:0.01mm.

3,G71/G72/G73 do Finishing Automatically [1 mean Yes,0 mean No]

It is set for when use G71/G72/G73 codes to processing, if do finishing automatically. 1:yes ; 0: No finishing.

4,(^u)X_Distance&Direction of finishing allowance of G71/G72/G73(10um)

It is set for every default remain of smoothing(diameter) in X-axis direction when it isn't be set in G71/G72/G73 code;unit:0.01mm.

5,(^w)Z_Distance&Ddirection of finishing allowance of G71/G72/G73(10um)

It is set for   every default remain of smoothing(diameter) in Z-axis direction when it isn't be set in G71/G72/G73 code;unit:0.01mm.

5-1, Auto_Feeding/Escaping Direction of G71/G72/G73[0:No,1:Yes]

It sets whether automatically judge feeding/escaping direction of G71/G72/G73 commands, 0: No ; 1: Yes, automatically judge.

6,(d)The number of division in G73

It is for default cycle times when it isn't be set in G73 code.

7,(^i)X_Distance and direction of relief of G73[Radius Designation](10um)

It is for default rough thickness of X axis when it isn't be set in G73 code.

8,(^k)Z_Distance and direction of relief of G73(10um)

It is for default rough thickness of Z axis when it isn't be set in G73 code.

9,(^k)Deptch of Cut in Z direction of G74/G75(10um) [Without Sign]

It is for each default infeed in Z-axis direction when it isn't be set in G74;

It is for every default infeed (radius) in X-axis direction when it isn't be set in G75 code;

10,(e)Return Amount of G74/G75(10um) [Radius Designation]

It is for every default retract in Z-axis direction when it isn't be set in G74 code;

It is for   every default retract (radius) in X-axis direction when it isn't be set in G75

code;unit:0.01mm.

11,(m)Repetitive Count in Finishing of G76

It is for default cycle times when it isn't be set in G76 code.(times:1-99)

12,(r)Chamfering amount of G76(1/10 lead)

It is for default length of retract chamfer when it isn't be set in G76 code.the length is 1/10 of thread lead.

13,(a)Angle of tool tip in G76(degree) [0~180°]

It is for default angle of thread tooth when it isn't be set in G76 code.unit:degree.

14,(^dmin)Minimum Cutting Depth of G76(10um) [Radius Designation]

It is for set minimal cutting depth (radius) of G76.Unit:0.1mm

15,(d)Finishing allowance of G76(10um)

It is for default remaining of finish turn when it isn't be set in G76 code.

16,Programming mode of X axis [1 mean Radius,0 mean Diameter]

There are two programming modes ,when it set as 1 that means radius programming mode,when set as 0 means that diameter programming mode

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 1means that running program with running spindle;when set as 0 means that running program without check spindle running.

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

It is for interlock between run program and spindle chuck is tightening, when set as 1 means that running program with chuck is tightening, input point is M14, PIN24_CN10 plug;when set as 0 means that running program without check condition of chuck.

18,Times of auto-cycle running(M20)

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

20,Length of chamfering in G92(1/10 lead)

It is for set default length(width) of quit and retract,the length=thread lead * 0.1.

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

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

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

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

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

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

24,Q(△ d)infeed times in G76 [8 mean times of rough cutting]

It is for set the define of Q in G76,set as "8",it is the times of feeding in roughing.

33,Type of Detect SP_Speed Reached(0:M69 Realy, 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 screensaver [>=2minutes]

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

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

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

203,Inspect overcut when tool radius offset (2131970:Yes, 34818:No,6326274:Pre-Read)

It is set for if inspect overcut when tool radius offset(G41/G42), when set to 2131970: yes, inspect; when set to 34818,don't inpect; when set 6326274,fast pre-read blocks.

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

It set the type of graphic display area, when set to 8, the graphic display area is set manually,& related parameter; when set to others, cnc system will adjust graphic display area automatically.

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

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

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

CNC system support run processing program by input points , +4+8+16+32+64+128, also D2-D7[2^2-2^7]: related input points is X26-X31, +256+512+1024+2048+4096+8192+16384 +32768 , also D8-D15 [2^8-2^{15}] : related input points is X16-X23.

Example: When P230=+4+8=12, inputs of X26 or X27 is valid, CNC system will running program of "X26"/"HIDEFILEX26" or "X27"/ "HIDEFILEX27".

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

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

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

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

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

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

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

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

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

2.2 Speed parameter

P	Parameter	EX-Value
1	G00 Speed of X-axis (mm/min)	6000.000
2	G00 Speed of Z-axis (mm/min)	6000.000
3	Manual Max Feeding Speed(mm/min)	10000.000
4	Auto Max Feeding Speed(mm/min)	15000.000
5	Default Speed of G01/G02/G03 (mm/min)	2000.000
6	Running Speed at Simulation Mode (mm/min)	10000.000
7	Manual Feeding Speed(mm/min)	100.00
8	Manual Spindle Speed (rpm)	200.00
11	Limit G01/G02/G03 Speed of each axis [1:Yes, 0:No]	0
12	Max Speed of X_G1/G2/G3 (mm/min)	2000
13	Max Speed of Z_G1/G2/G3 (mm/min)	2000
14	X_Acceleration/Deceleration Constant [1~99999]	50000
15	Z_Acceleration/Deceleration Constant [1~99999]	50000
16	Acceleration/Deceleration Constant When Auto Run	2
17	Handwheel_Acce/Dece-leration Constant [500--32000]	600
18	Handwheel_Acce/Dece-leration Constant when dry [>500]	600
19	G00 Speed when dry (mm/min) [>10]	1500
20	X_Max Speed with Handwheel (mm/min)	2000
21	Z_Max Speed with Handwheel (mm/min)	2000
22	Z_Acceleration/Deceleration Constant when threading	0.000
23	X_Acceleration/Deceleration Constant when threading	0.000
24	Ratio of Retired Speed in X-axis Servo when threading	5000
25	Beginning Speed of Retired in X-axis Step when threading	150
26	Max Retired Speed in X-axis Step when threading	6000
27	Type of Acceleration/Deceleration [0 mean line,8 mean curve]	0
28	Initial Acce/Dece-leration Constand when curve [>=10]	100
29	Quadratic Acce/Dece-leration Constant when curve [>=10]	500
30	Max Acce/Dece-leration Constant when curve [>=500]	1
31	X_Homing Speed (mm/min)	2500.000
32	X_Speed for detect Z0 signal (mm/min)	250.000
33	Z_Homing Speed (mm/min)	2500.000
34	Z_Speed for detect Z0 signal (mm/min)	250.000
35	Min Speed of spindle when G96 (rpm)	1
36	Max Speed of 1st Spindle (rpm)	3000
37	Max Speed of Spindle at 2nd gear (rpm)	3000
38	Max Speed of Spindle at 3rd gear (rpm)	3000
39	Max Speed of Spindle at 4th gear (rpm)	3000
40	Max Speed of 2nd Spindle (rpm)	3000
41	Compensation Mode of Arc Backlash (0: A, 8: B)	8
42	Compensation Speed in B Mode (mm/min)	3000
42-1	Beginning Compensation Speed in B Mode (mm/min)	500
42-2	Acce/Dece-leration Constant in B mode (mm/min/s)	60000
43	Handwheel_Stopping Speed (mm/min)[>100]	100
49	Activate Speed Processing Function[1:Yes, 0:No]	0
50	Handwheel Stop Speed(mm/min)[>100]	
51	SP_Speed when exit at following tapping mode(rpm)[>1]	100
52	SP_Reverse Backlash Compensation when tapping(Pulse)	1
53	Advance Retired Value before reverse rotation of following tapping(um)[10-5000]	1
54	Retired Speed when tapping, (mm/min)[>=60]	1
58	Beginning Speed when hardware limitation (mm/min)	1.000
60	Use smooth processing function(+4:Mauanl,+8:MPG,+16:Program)	60

61	Time constant of smooth processing on Manual[2-500]	50
62	Time constant of smooth processing on MPG[2-500]	
63	Time constant of smooth processing on Program[2-500]	
101	X_Beginning Speed (mm/min)[>1]	500
102	Y_Beginning Speed (mm/min)[>1]	500
103	Z_Beginning Speed (mm/min)[>1]	500
104	A_Beginning Speed (mm/min)[>1]	500
105	B_Beginning Speed (mm/min)[>1]	500
111	X_Max Speed Increment (mm/min)[>1]	500
112	Y_Max Speed Increment (mm/min)[>1]	500
113	Z_Max Speed Increment (mm/min)[>1]	500
114	A_Max Speed Increment (mm/min)[>1]	500
115	B_Max Speed Increment (mm/min)[>1]	500
200	Coherent movement is valid for G00[1:No,16:Yes]	1
201	Wait SP_Speed smooth when threading	0

Explanation about Speed Parameter:

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

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

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

3,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.

4,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.

5,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)

6,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)

7,Manual Feeding Speed(mm/min)

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

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

8,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 paramter directly.

11,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.

12,Max Speed of X_G1/G2/G3 (mm/min)
      It is for the Max running speed of X-axis when G1/G2/G3 interpolation.

13,Max Speed of Z_G1/G2/G3 (mm/min)
      It is for the Max running speed of Z-axis when G1/G2/G3 interpolation.

14,X_Acceleration/Deceleration Constant   [1~99999]
      It is time constant of X-axis acc/dec-eleration,the bigger it is ,the faster the ace/dec-eleration is.
    Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

15,Z_Acceleration/Deceleration Constant   [1~99999]
      It is time constant of Z-axis acc/dec-eleration, the bigger it is ,the faster the ace/dec-eleration is.
    Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

16,Acceleration/Deceleration Constant When Auto Run [1-500]
      It is for set constant of acc/dec-eleration in auto.the range is 1-500.It is mainly for distinguish Auto and Manual,only the difference is too much,set it is effective.

17,Handwheel_Acce/Dece-leration Constant   [500--32000]
      It is for set constant of acc/dec-eleration of Handwheel.the range is 500-32000.
18,Handwheel_Acce/Dece-leration Constant when dry [>500]
      It is for set constant of acc/dec-eleration when handwheel start program. the range is 500-32000.when the value is less than 500,it is invalid.

19,G00 Speed when dry (mm/min)   [>10]
      It is the speed of G00 when when handwheel start program for simulate. it is invalid when the value is less than 10.

20,X_Max Speed with Handwheel (mm/min)
      It is for limit max speed of X-axis when use handwheel in manual.
    Attention:it is valid when >100,otherwise invalid.And suggest don't over 4000.

21,Z_Max Speed with Handwheel (mm/min)
      It is for limit max speed of Z-axis when use handwheel in manual.
    Attention:it is valid when >100,otherwise invalid.And suggest don't over 4000.

22,Z_Acceleration/Deceleration Constant when threading

It is time constant of Z-axis ace/dec-eleration when do threading , Range:300~90000.The bigger it is,the faster the ace/dec-elecration.

Attention:with stepper system,it should be set smaller. When <300,it is invalid.When with servo motor ,for ensure efficiency,set 0.

23,X_Acceleration/Deceleration Constant when threading

It is time constant of Z-axis ace/dec-eleration when do threading, the range is from 300 to 90000. The bigger it is,the faster the ace/dec-elecration.

Attention:with stepper system,the smaller,the safer.when <300, it is invalid. When with servo motor, for ensure efficiency, set 0.

24,Ratio of Retired Speed in X-axis Servo when threading

It is ratio of X axis retired speed that configured with servo system when threading. Unit: *0.1 .

25,Beginning Speed of Retired in X-axis Step when threading

It is beginning speed of X axis retired that configured with step system when threading.

Attention: for safe,it should be less than 100mm/min.

26,Max Retired Speed in X-axis Step when threading

It is Max speed of X axis retired that configured with step system when threading.

Attention: it is related to load,the bigger the load,the samller it is.

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

It sets type of acce/dece-leration.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.

28,Initial Acce/Dece-leration Constand when curve [>=10]

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

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

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

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

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

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

31,X_Homing Speed (mm/min)

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

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

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

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

33,Z_Homing Speed (mm/min)

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

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

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

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

35,Min Speed of spindle when G96 (rpm)

It is the min speed of spindle when run G96 code , also at constant surface speed mode.

36,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).

37,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.

38,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.

39,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.

40,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.

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

It is compensation mode for arc reverse backlash.

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

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

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

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

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

42-1,Beginning Compensation Speed in B mode (mm/min)
It is beginning compensatin speed in B compensation mode.it is valid when it >10.

42-2,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.

43,Handwheel_Stopping Speed (mm/min)[>100]
It is the speed when handwheel stop. the bigger it is ,the faster handwheel stop.

49,Activate Speed Processing Function[1:Yes ; 0:No]
It is set for if activate speed processing function, 1 means yes, activate speed processing function, 0 means no activate the function.

50,Handwheel Stop Speed(mm/min)[>100]
It sets the starting speed that handwheel stop.

51,SP_Speed when exit at following tapping mode
It is min speed before spindle reverse ratation when tapping.

52,SP_Reverse Backlash Compensation when tapping
It is reverse backlash compensation value before spindle reverse rotation when tapping. Unit: Pulse

53,Advance Retired Value before reverse rotation of following tapping
It is advanve retired value before spindle reverse rotation when tapping. Unit:um. Range:10-5000

54,Retired Speed when tapping (mm/min)[>=60]
It is speed when spindle retired during tapping. Unit: mm/min

58,Beginning Speed when hardware limitation (mm/min)
It is beginning speed that motor touch harware limitation switch. When cnc system is configured with servo, no need to set beginning speed , set to 1 normally.

60,Use smooth processing function(+4:Manual,+8:MPG,+16:Program)
It is set mode of using smooth processing function,+4: smooth processing on Manual; +8: smooth processing on MPG(Handwheel); +16: smooth processing on running program.

61,Time constant of smooth processing on Manual[2-500]
62,Time constant of smooth processing on MPG[2-500]
63,Time constant of smooth processing on Program[2-500]

It is related time constant when cnc system uses smooth processing function, Range: 2-500.

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]

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]

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

Example:when it is 300,the speed of X axis(multiaxial 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.

201,Wait SP_Speed smooth when threading[0:No, 1:Yes]

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

2.3 Axis parameter

P	Axis Parameter	Ex-Value
1	Switch Type for Feed-Rate [0: Key, 1: Band Switch]	0
2	Switch Type for SP-Rate [0: Key, 1: Band Switch]	0
3	Max Travel in X_Negative direction (mm)	-9999.000
4	Max Travel in X_Positive direction (mm)	9999.000
5	Max Travel in Z_Negative direction (mm)	-9999.000
6	Max Travel in Z_Positive direction (mm)	9999.000
7	SP_Braking Time (10ms)	150
8	SP_Braking is Long Signal [0: No,1: Yes]	0
9	Detect SP_Position Feedback [0: No,1: Yes]	1
10	Pulses Per Revolution of Spindle	4096
10-1	Allow Error of Resolution of SP_Encoder[>10]	
10-2	Pulses Per Revolution of Spindle[1/1:0,Other:>99]	
11	Soft-Limitation is valid [0:Yes, 1: No][D2X;D3C(Y);D4Z;D5A;D6B]	0
12	X_Reverse Backlash Compensation (um)	00000001
13	Z_Reverse Backlash Compensation (um)	0
14	X_Direction [1:normal, 0: Reverse]	0
15	Z_Direction [1:normal, 0: Reverse]	0
16	Using Electron Gear Ratio for Feeding Axes [0:Yes, 1:No]	0
17	Numerator of X_Electron Gear	1
18	Denominator of X_Electron Gear	1
19	Numerator of Z_Electron Gear	1
20	Denominator of Z_Electron Gear	1
21	Type of Limit Switch in Positive Direction[0:NO type, 1:NC type]	0
22	Type of Limit Switch in Negative Direction[0:NO type, 1:NC type]	0
23	Type of Home [D3X;D4(C)Y;D5Z;D6A;0:Switch;1:float Zero]	01111011
24	X_Machine Coordinate of float zero point	
25	Z_Machine Coordinate of float zero point	
26	Grade of Homing[1:No,0:Prompt,8:Compulsion,9:Super compulsion]	1
27	Mode of Homing	1
28	Direction of Homing[D2:X,D3:C(Y),D4:Z,D5:A,D6:B]	0000000100000001
29	Type of Switchs for Homing	00000000
30	Range of Detect Z0 in X axis	45
31	Range of Detect Z0 in Z axis	60
32	Offset after homing in X axis	0
33	Offset after homing in Z axis	0
50	Spindle is rotating when shift gear[1:Yes, 0:No]	0
51	Rotating Speed of Spindle when shift gear(1/100rpm)	100
52	Rotating Direction of Spindle when shift gear[0:CW, 1:CCW]	1
53	Pause Time of Spindle when shift gear (10ms)	10
54	Braking Time of Spindle when shift gear (10ms)	10
55	Delay time between reset M03/M04 & set M05 (10ms)	0
68	Delay Time when feeding axes shift direction (ms)	0
80	X/Z axis is Rotating Axis	00000001
100	System Inner Parameter	1
101	Name of 3rd Axis [0:Y, 1:C]	1
102	Mode of Y(C) Axis [0:Rotating Axis, 1: Linear Axis]	-9999.000
104	C(Y)_Direction [1:normal, 0: Reverse]	9999.000
105	Numerator of C(Y)_Eletronic Gear	1

106	Denominator of C(Y)_Eletronic Gear	1
107	C(Y)_Reverse Backlash Compensation (um)	0
108	G00 Speed of C(Y) Axis	4000.000
109	Max Speed of C(Y)_G1G2G3	2000
110	C(Y)_Acceleration/Deceleration Constant	50000
111	C(Y)_Max Speed with Handwheel (mm/min)	2000
112	Speed of C-axis return to Zero point of Encoder(°/min)	250.000
113	Y_Homing Speed (mm/min)	2500.000
114	Y_Speed for Detect Z0 signal (mm/min)	250.000
115	Range of detect Z0 in Y axis	60
116	Offset after homing in Y axis	0
117	Max Travel in C(Y)-Negative Direction	-9999.000
118	Max Travel in C(Y)-Positve Direction	9999.000
119	C(Y)_Machine Coordinate of float zero point	
200	System Inner Parameter	0
201	Mode of A Axis [0:Rotating Axis, 1: Linear Axis]	1
202	Base when A axis is rotating axis	0
203	A_Direction [1:normal, 0: Reverse]	0
204	Numerator of A_Eletronic Gear	1
205	Denominator of A_Eletronic Gear	1
206	A_Reverse Backlash Compensation (um)	0
207	G00 Speed of A Axis	4000.000
208	Max Speed of A_G1G2G3	2000
209	A_Acceleration/Deceleration Constant	50000
210	A_Max Speed with Handwheel (mm/min)	2000
211	Homing Speed in A Positive Direction (mm/min)	2500.000
212	Homing Speed in A Negative Direction (mm/min)	250
213	Range of detect Z0 in A axis	60
214	Offset after homing in A axis	0
215	Max Travel in A-Negative Direction	-9999.000
216	Max Travel in A-Positve Direction	9999.000
217	A_Machine Coordinate of float zero point	
301	Mode of B Axis [0:Rotating Axis, 1: Linear Axis]	1
302	Base when B axis is rotating axis	0
303	Name of 5th Axis [0:B, 1:C]	0
304	B_Direction [1:normal, 0: Reverse]	0
305	Numerator of B_Eletronic Gear	1
306	Denominator of B_Eletronic Gear	1
307	B_Reverse Backlash Compensation (um)	0
308	G00 Speed of B Axis	4000.000
309	Max Speed of B_G1G2G3	2000
310	B_Acceleration/Deceleration Constant	50000
311	B_Max Speed with Handwheel (mm/min)	2000
312	Homing Speed in B Positive Direction (mm/min)	2500.000
313	Homing Speed in B Negative Direction (mm/min)	250
314	Range of detect Z0 in B axis	60
315	Offset after homing in B axis	0
316	Max Travel in B-Negative Direction	-9999.000
317	Max Travel in B-Positve Direction	9999.000
318	B_Machine Coordinate of float zero point	
404	SP_Direction when position control mode	0

405	Using Eletronic Gear Ratio for Spindle [0:Yes, 1:No]	0
406	Numerator of SP_Eletronic Gear Ratio in Low Gear	4096
407	Denominator of SP_Eletronic Gear Ratio in Low Gear	36000
408	Numerator of SP_Eletronic Gear Ratio in High Gear	4096
409	Denominator of SP_Eletronic Gear Ratio in High Gear	360000
410	Coordinate Axis when spindle do tapping	92
411	Control Mode of Tapping [0:Following, 4: Interpolation]	0
412	Teeth of SP_Motor (<P413)	1
413	Teeth of SP_Encoder (>P412)	1
414	Follow-Up of A Axis[7:X, 8:Y, 9:Z]	1

Explanation about Axis Parameter:

◆ **P81-P400 are the parameters for C(Y)axis, A axis & B axis (only when cnc system is configured with C(Y) , A axis & B axis, related parameters are effective), take X-axis/Z-axis as reference.**

1,Switch Type for Feed-Rate [0: Key, 1: Band Switch]
It is set switch type of Feed-Rate, Rate of Feeding axes.
0: Keys of "Feed Rate+" & "Feed Rate-" in Panel for adjust rate of feeding speed;
1: External Band switch for Feed-Rate. Port for external band switch is at CN11 plug, Pins are VDK0,VDK1,VDK2,VDK3. Total are 16 gears

2,Switch Type for SP-Rate   [0: Key, 1: Band Switch]
It is set switch type of SP-Rate, Rate of SP_Speed.
0: Keys of "SP Rate+" & "SP Rate-" in Panel for adjust rate of spindle speed;
1: External Band switch for SP-Rate. Port for external band switch is at CN11 plug, Pins are VDS0,VDS1,VDS2,VDS3. Total are 16 gears

3,Max Travel in X_Negative direction (mm)
It is max travel in negative direction of X axis when soft-limitation, which is based on machine coordinate system.

4,Max Travel in X_Positive direction (mm)
It is max travel in positive direction of X axis when soft-limitation, which is based on machine coordinate system.

5,Max Travel in Z_Negative direction (mm)
It is max travel in negative direction of Z axis when soft-limitation, which is based on machine coordinate system.

6,Max Travel in Z_Positive direction (mm)
It is max travel in positive direction of Z axis when soft-limitation, which is based on machine coordinate system.

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

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

9,Detect SP_Position Feedback [0: No,1: Yes]
It is for whether the system detect position feedback signal of spindle by SP_encoder. 1: detect,

40