In this user manual we have tried to describe the matters concerning the operation of this CNC system to the greatest extent. However, it is impossible to give particular descriptions for all unnecessary or unallowable operations due to length limitation and products application conditions; Therefore, the items not presented herein should be regarded as “impossible” or “unallowable”.

Preface

Your Excellency,

We are honored by your purchase of this GSK988TA/988TA1/988TB Turning CNC System made by GSK CNC Equipment Co., Ltd.

This manual is the “Installation & Debugging” (Software Version: 1.12) of GSK988TA/GSK988TA1/GSK988TB Lathe CNC System User Manual, which is detailed the proceedings such as the installation and debugging in this User Manual.

Note: This manual is described based upon the GSK988TA.

To ensure safe and effective running, please read this manual carefully before installation and operation.

Warning

Accident may occur by improper connection and operation!

This system can only be operated by authorized and qualified personnel.

Special caution:

The power supply fixed on/in the cabinet is exclusively used for the CNC system made by GSK.

It can't be applied to other purposes, or else it may cause serious danger!

Cautions

■ Delivery and storage

- Packing box over 6 layers in pile is unallowed.
- Never climb the packing box, stand on it or place heavy objects on it.
- Do not move or drag the products by the cables connected to it.
- Forbid collision or scratch to the panel and display screen.
- Avoid dampness, insolation and drenching.

■ Open-package inspection

- Confirm that the products are the required ones.
- Check whether the products are damaged in transit.
- Confirm that the parts in packing box are in accordance with the packing list.
- Contact us in time if any inconsistence, shortage or damage is found.

■ Connection

- Only qualified personnel can connect the system or check the connection.
- The system must be earthed, and the earth resistance must be less than 0.1Ω. The earth wire cannot be replaced by zero wire.
- The connection must be correct and firm to avoid any fault or unexpected consequence.
- Connect with surge diode in the specified direction to avoid damage to the system.
- Switch off power supply before plugging out or opening electric cabinet.

■ Troubleshooting

- Switch off power supply before troubleshooting or changing components.
- Check the fault when short circuit or overload occurs. Restart can only be done after troubleshooting.
- Frequent switching on/off of the power is forbidden, and the interval time should be at least 1 min.

Announcement

- This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot be explained one by one due to so many possibilities that may involve with, so the contents that are not specially stated in this manual shall be considered as unallowable.

Warning

- Before installing, connecting, programming and operating, please carefully read the product user manual and the manual from the machine tool manufacturer and strictly operate accordance with the regulations in the manual; otherwise, the product or the machine tool may be damaged, the workpiece may get rejected, even the personal injury may occur.

Caution

- Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function deployment and technical performance of the machine tool are designed by the machine tool manufacturer, so function configuration and technical indexes are subject to the user manual from the machine tool manufacturer.

  Refer to the user manual from the machine tool manufacturer for function and meaning of each button on the machine panel.

All specifications and designs herein are subject to change without notice.

Safety Responsibility

Manufacturer's Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on design and configuration of the provided CNC systems and accessories.

——Be responsible for the safety of the provided CNC systems and accessories.

——Be responsible for the provided message and advice for the users.

User's Responsibility

——Be responsible for being familiar with and mastering the safety operation procedures through training with the safety operation of the CNC system.

——Be responsible for the dangers caused by adding, changing or altering the original CNC systems and the accessories.

——Be responsible for the dangers caused by failing to observe the provisions in the manual for operation, adjustment, maintenance, installation and storage.

This manual is kept by the end user.

Thank you for supporting us in the use of GSK's products!

List

BOOK I INSTALLATION & CONNECTION

Chapter 1 Installation Layout

1.1 Installation Dimension of GSK988TA/988TA1/988TB and its Accessory

GSK988TA/988TA1/988TB divid into GSK988TA1 (Vertival), GSK988TA1-H (Horizontal), GSK988TA (Vertival), GSK988TA-H (Horizontal), GSK988TB (10.4 inch vertical) and GSK988TB-H (10.4 inch horizontal), and its configured operation panels are also different, refer to the following table for the detailed types.

Production type	Panel name	Structure	Name
GSK988TA1 (vertical-type)	Machine operational panel	With MPG	MPU-08E
		Without MPG	MPU-09E
GSK988TA1-H (horizontal-type)	Machine operational panel	With MPG	MPU-10E
		Without MPG	MPU-11E
GSK988TA (vertical-type)	Machine operational panel	With MPG	MPU-08
		Without MPG	MPU-09
GSK988TA-H (horizontal-type)	Machine operational panel	With MPG	MPU-10
		Without MPG	MPU-11
GSK988TB (10.4 inch screen vertical-type)	Editing keyboard		EDU-01
	Machine operational panel		MPU-20
	Machine operational panel	With MPG	AP04
		Without MPG	AP05
GSK988TB-H (10.4 inch screen horizontal-type)	Editing keyboard		EDU-02
	Machine operational panel		MPU-20
	Machine operational panel	With MPG	AP06
		Without MPG	AP07

Table 1-1

1.1.1 GSK988TA1 and its Accessory

1.1.1.1 GSK988TA1 Host Figure Installation Dimension

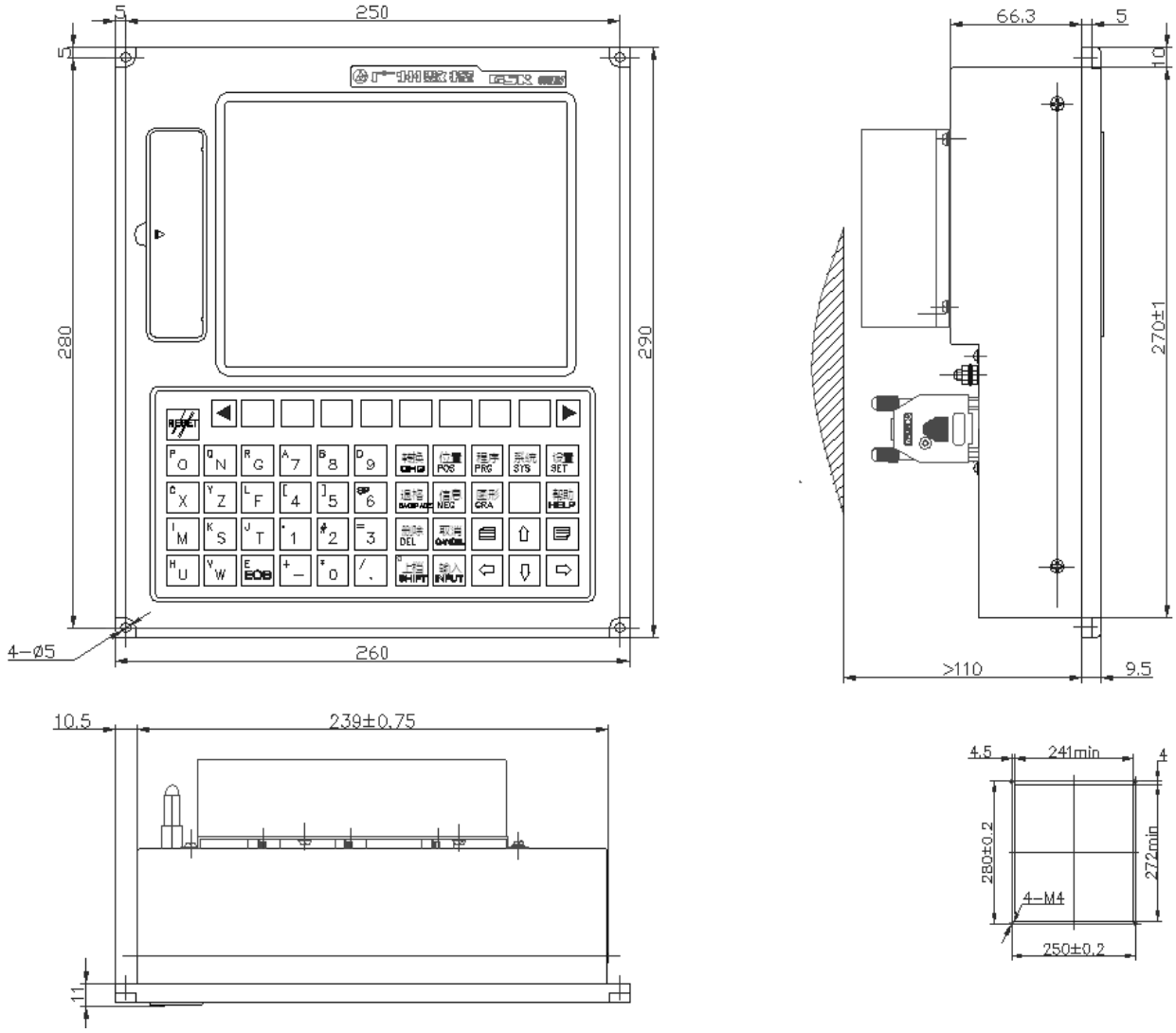


Fig. 1-1 GSK988TA1 appearance installation dimension

1.1.1.2 Outline Installation Dimension of GSK988TA1 Operation Panel MPU-08E

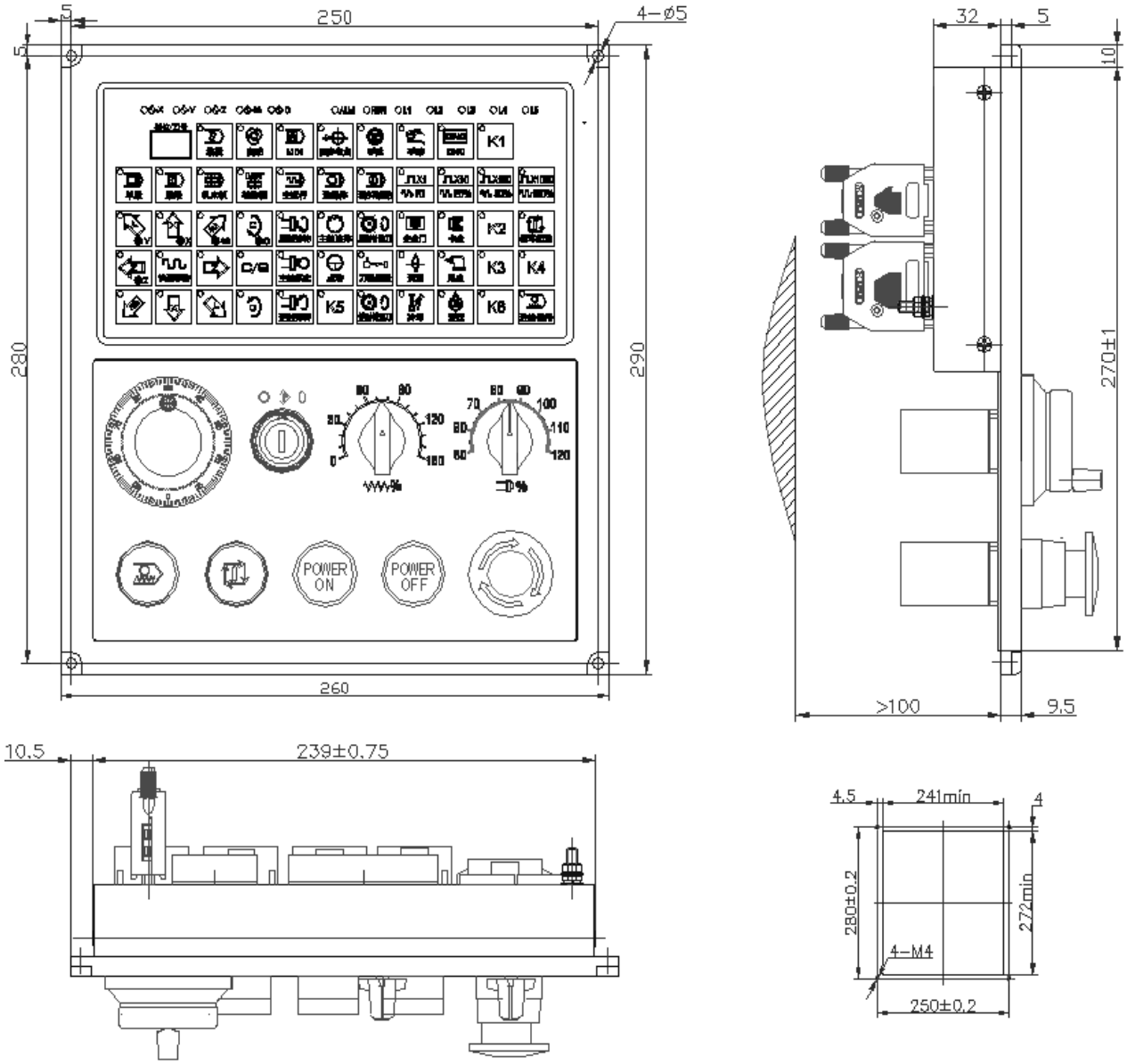


Fig. 1-2 The installation dimension of machine operation panel MPU-08E

Note: The installation dimension of the operation panel MPU-09E is identical with the one of the MPU-08E, which is the different between them is with or without MPG.

1.1.2 GSK988TA1-H & Accessory

1.1.2.1 GSK988TA1-H Host Appearance Installation Dimension

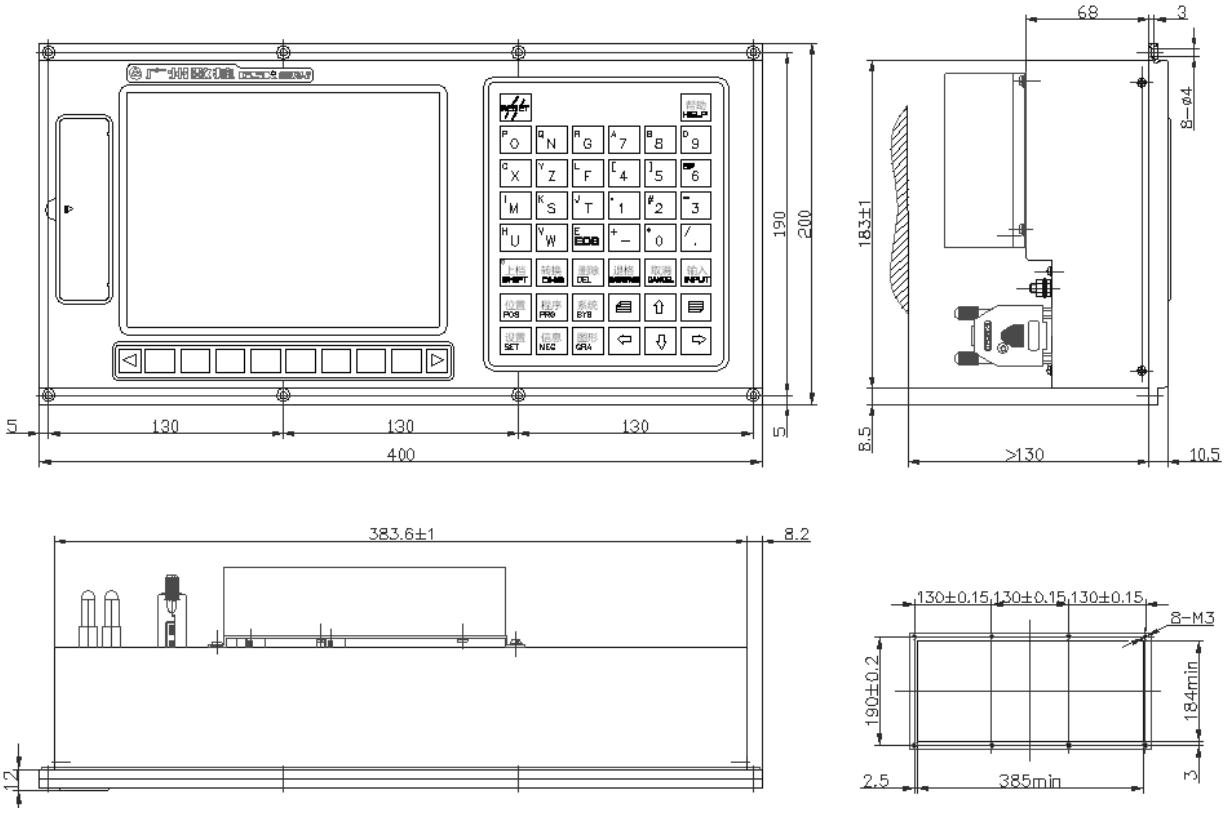


Fig.1-3

1.1.2.2 MPU-10E Appearance Installation Dimension of GSK988TA1-H Operation Panel

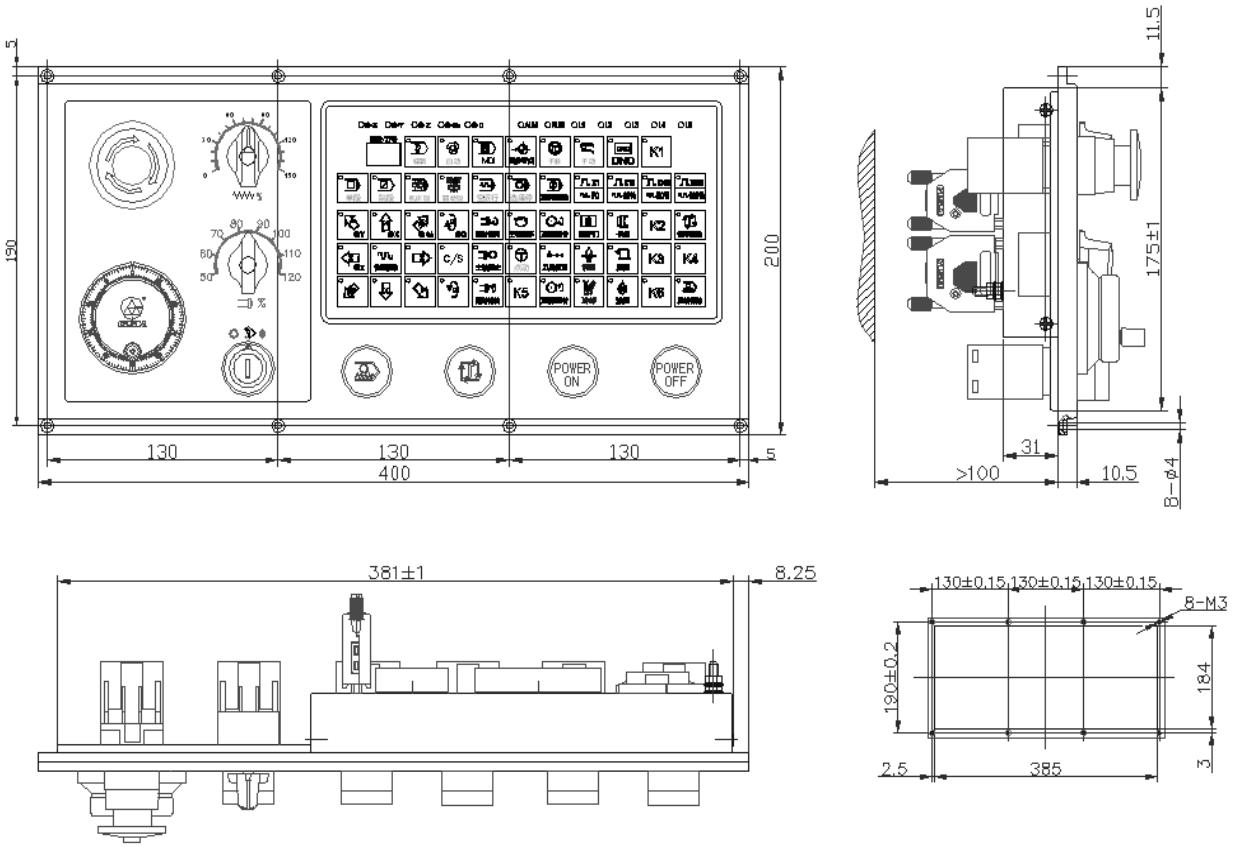


Fig.1-4

Note: The installation dimension of the operation panel MPU-10E is identical with the one of the MPU-11E, which is the different between them is with or without MPG.

1.1.3 GSK988TA and its Accessory

1.1.3.1 GSK988TA Host Figure Installation Dimension

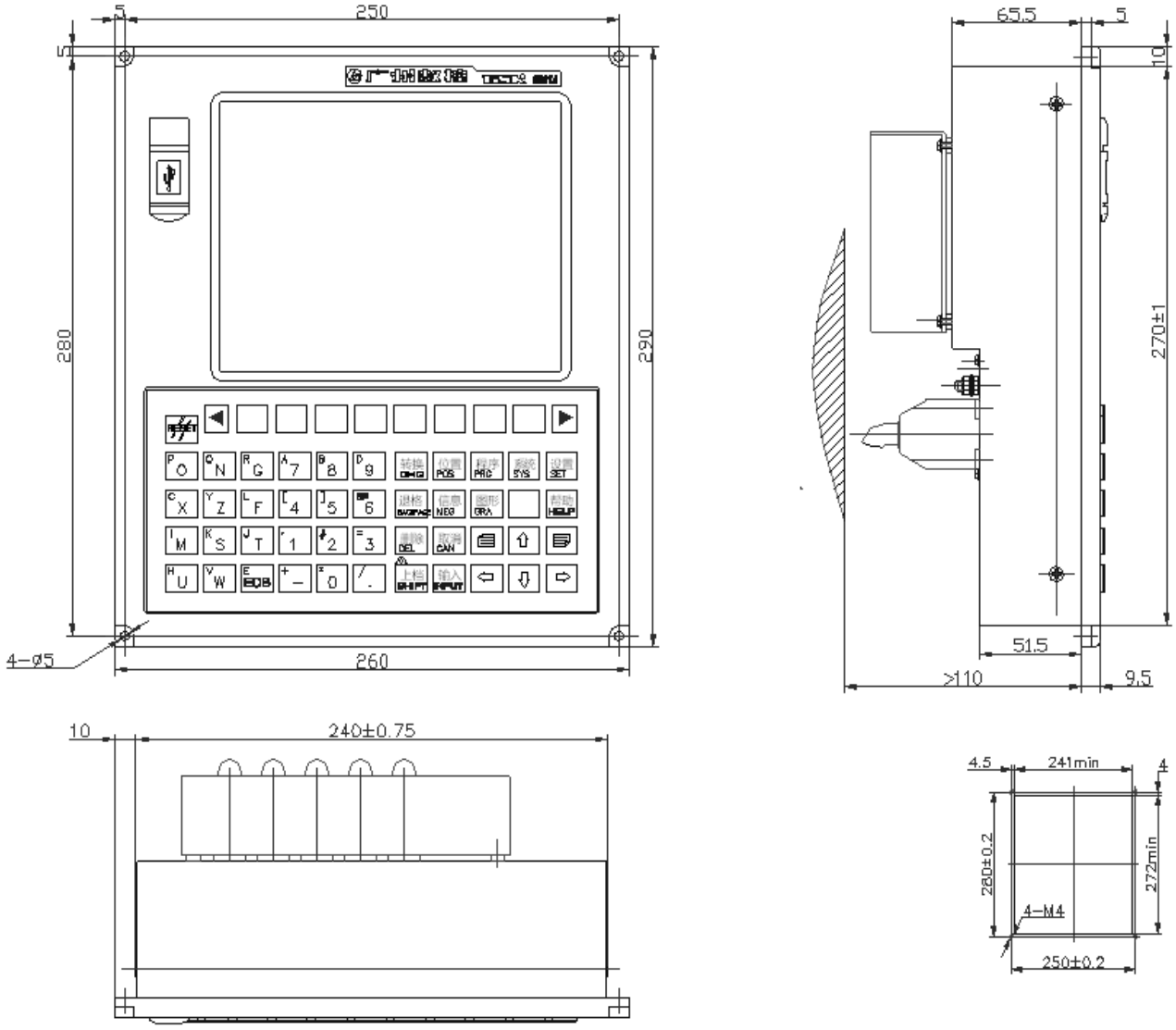


Fig.1-5

1.1.3.2 Appearance Installation Dimension of GSK988TA Operation Panel MPU-08

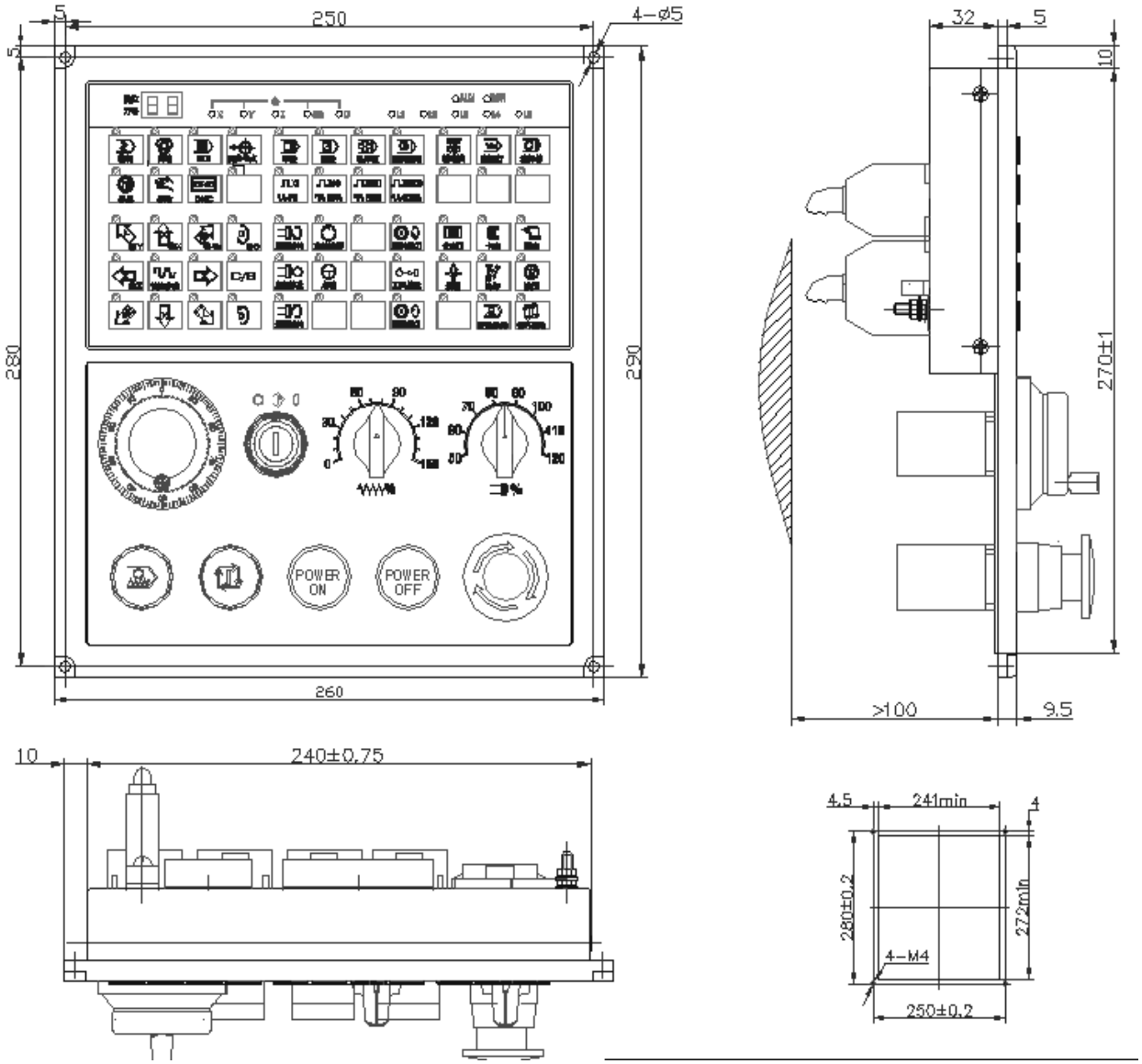


Fig.1-6

Note: The installation dimension of the operation panel MPU-09 is identical with the one of the MPU-08, which is the different between them is with or without MPG.

1.1.4 GSK988TA-H & Accessory

1.1.4.1 GSK988TA-H Host Appearance Installation Dimension

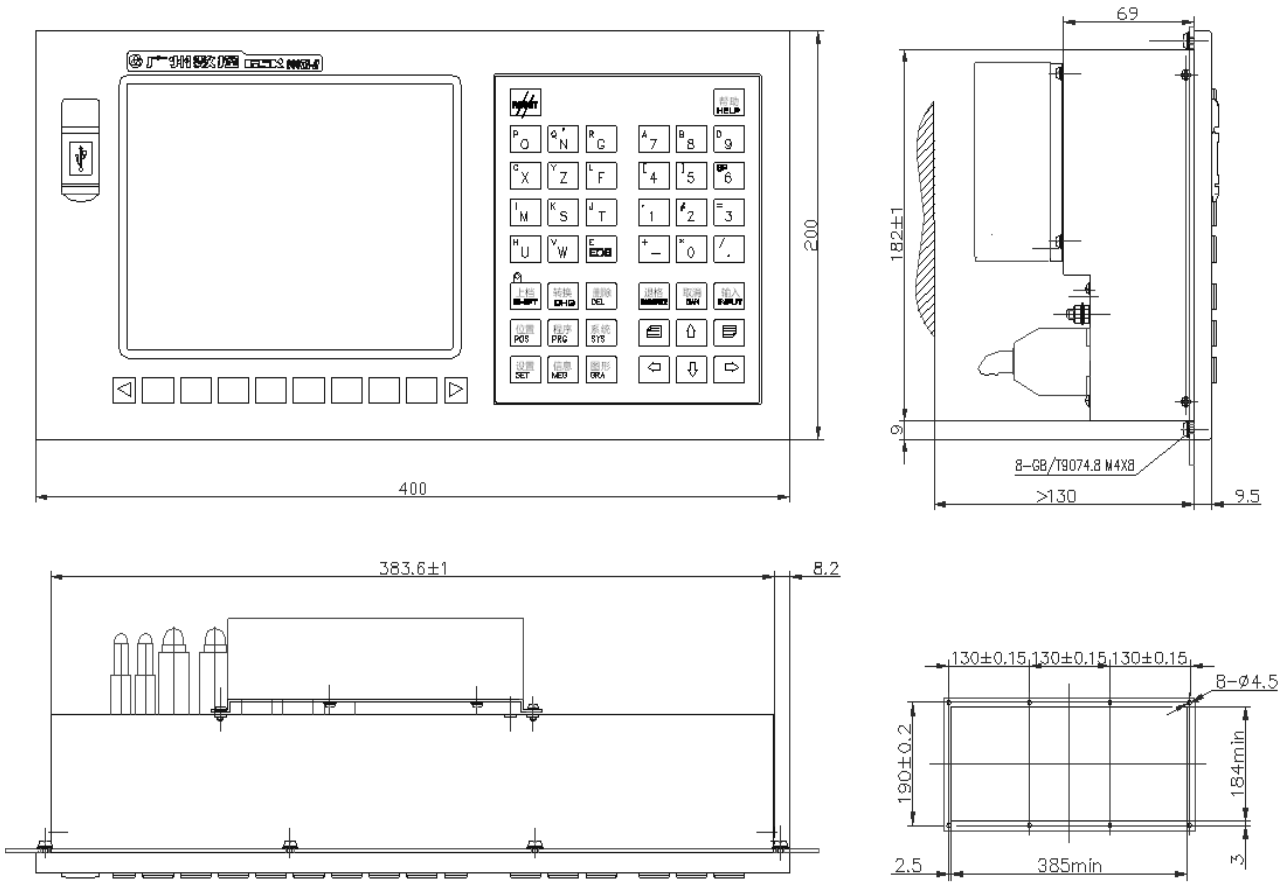


Fig.1-7

1.1.4.2 MPU-10 Appearance Installation Dimension of GSK988TA-H Operation Panel

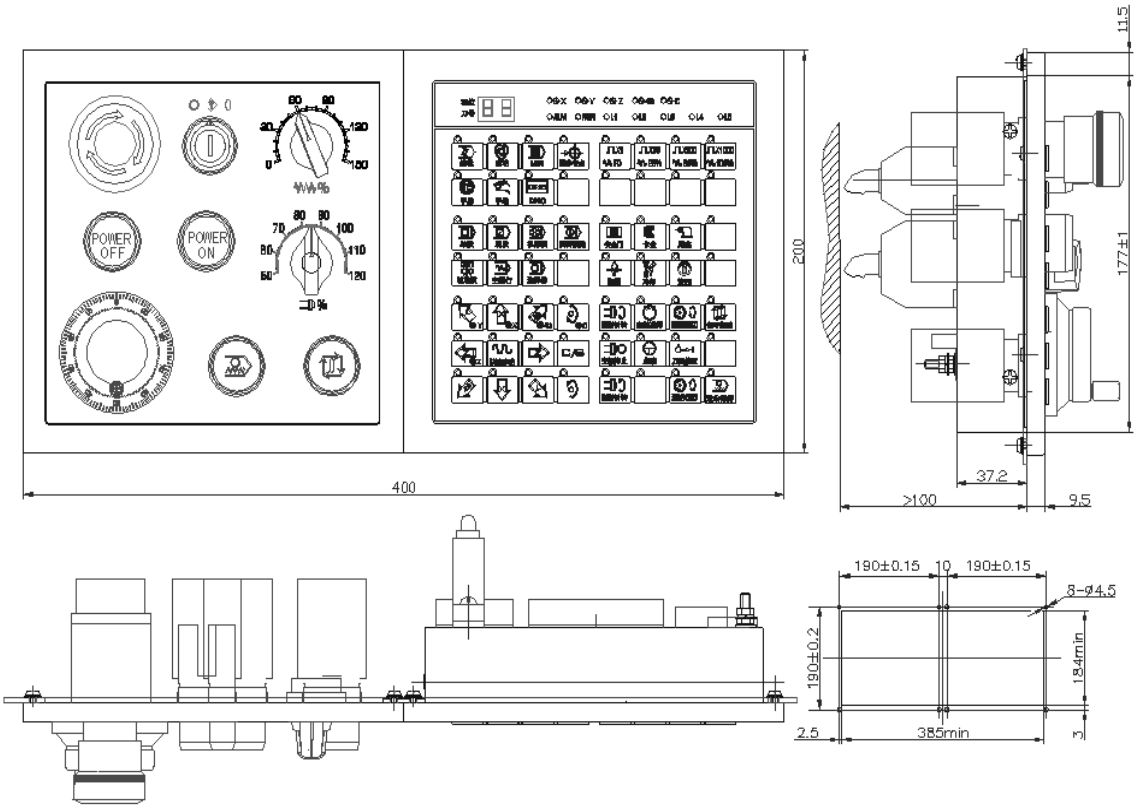


Fig.1-8

Note: The installation dimension of the operation panel MPU-10 is identical with the one of the MPU-11, which is the different between them is with or without MPG.

1.1.5 GSK988TB and its Accessory

1.1.5.1 GSK988TB Host Outline Installation Dimension

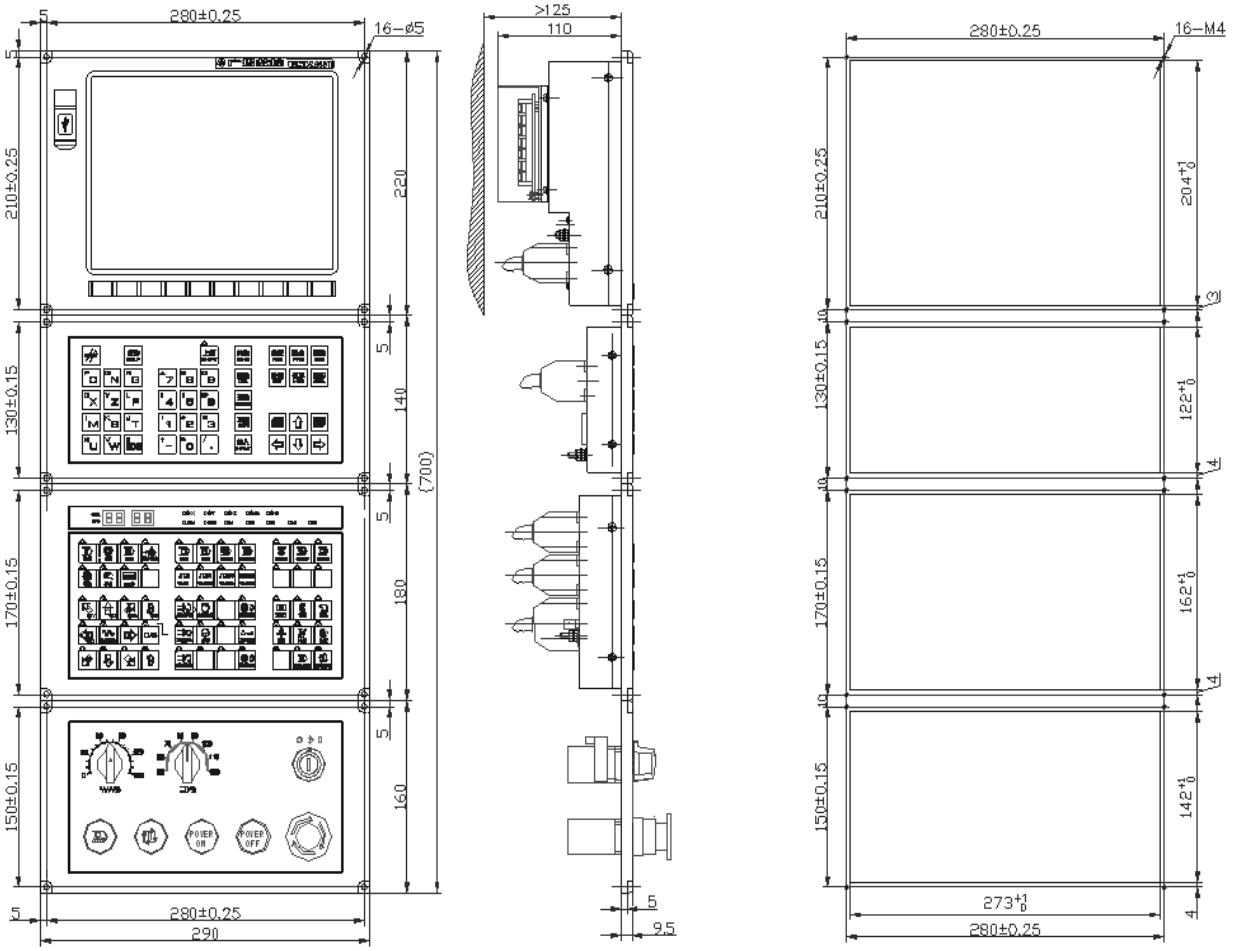


Fig. 1-9

1.1.5.2 GSK988TB-H Host Outline Installation Dimension

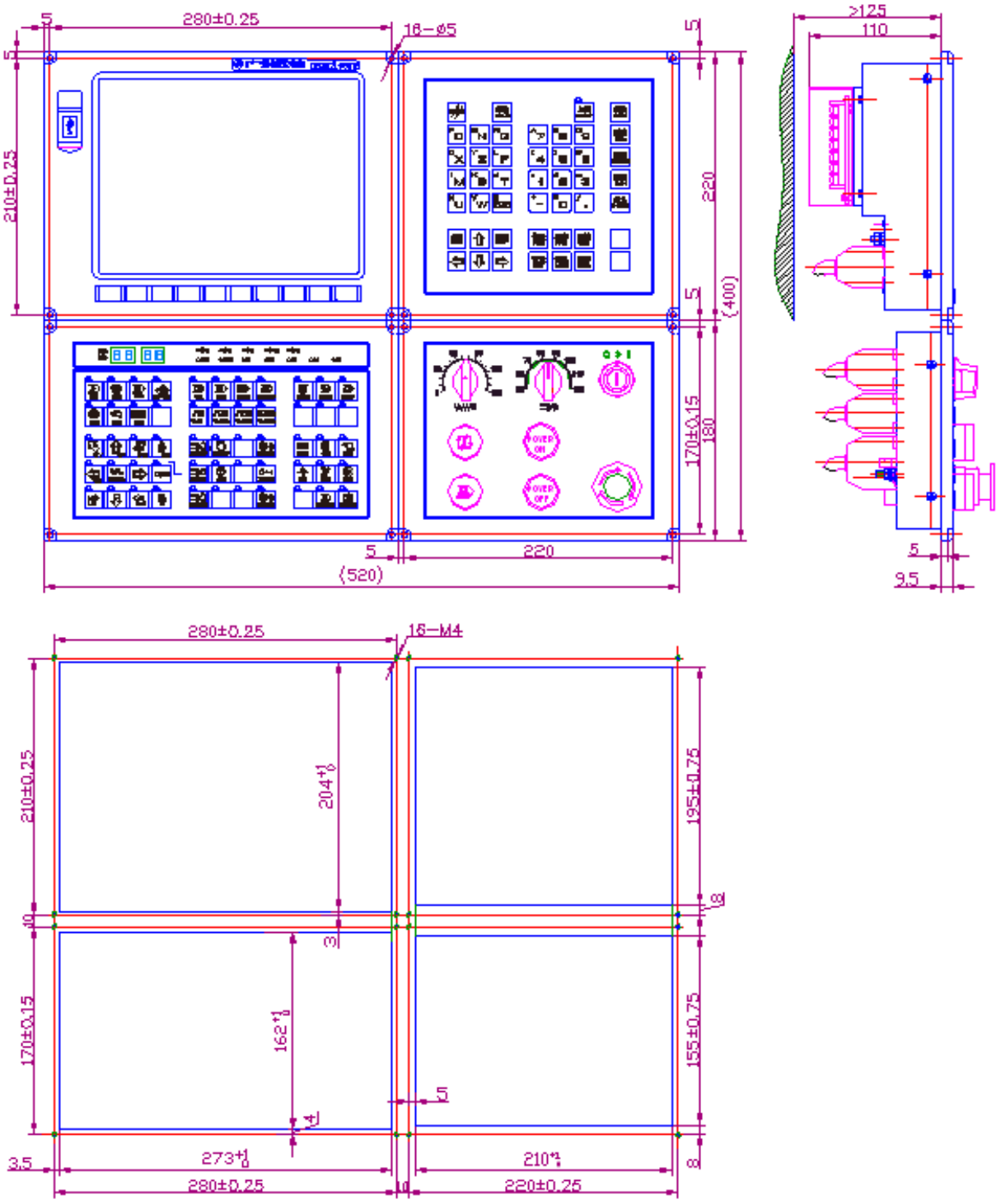


Fig. 1-10

1.1.6 I/O Unit Appearance Dimension

1.1.6.1 IOL-01T Appearance Dimension

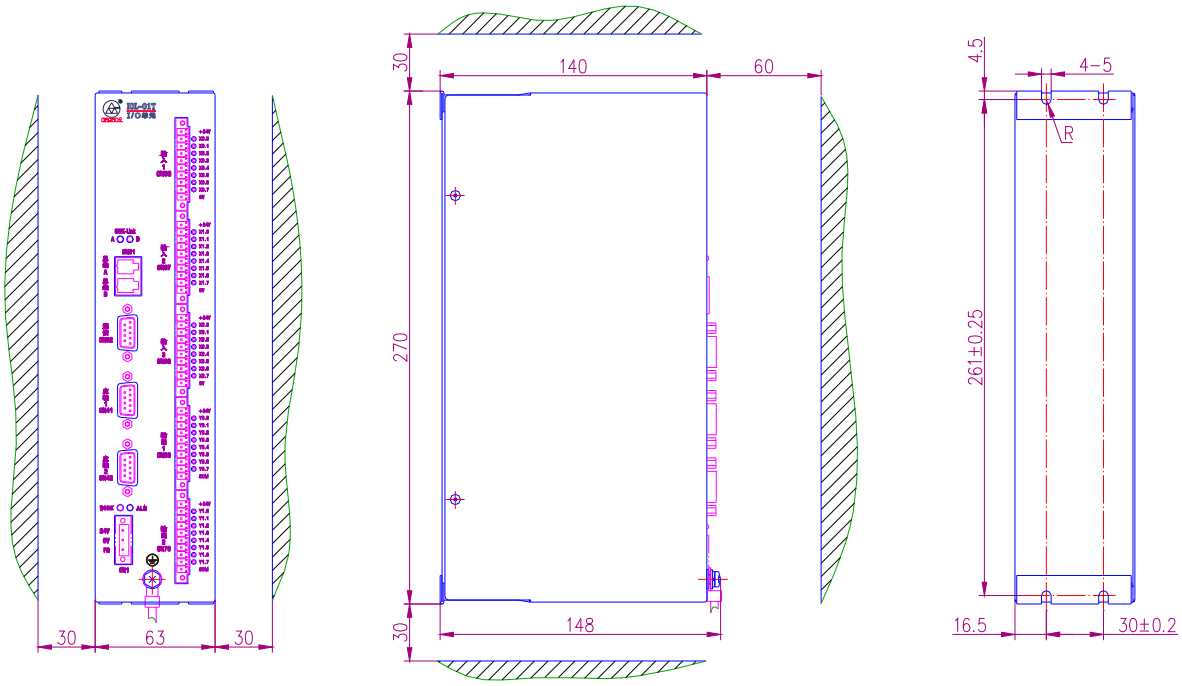


Fig. 1-11

1.1.6.2 IOL-02T Appearance Dimension

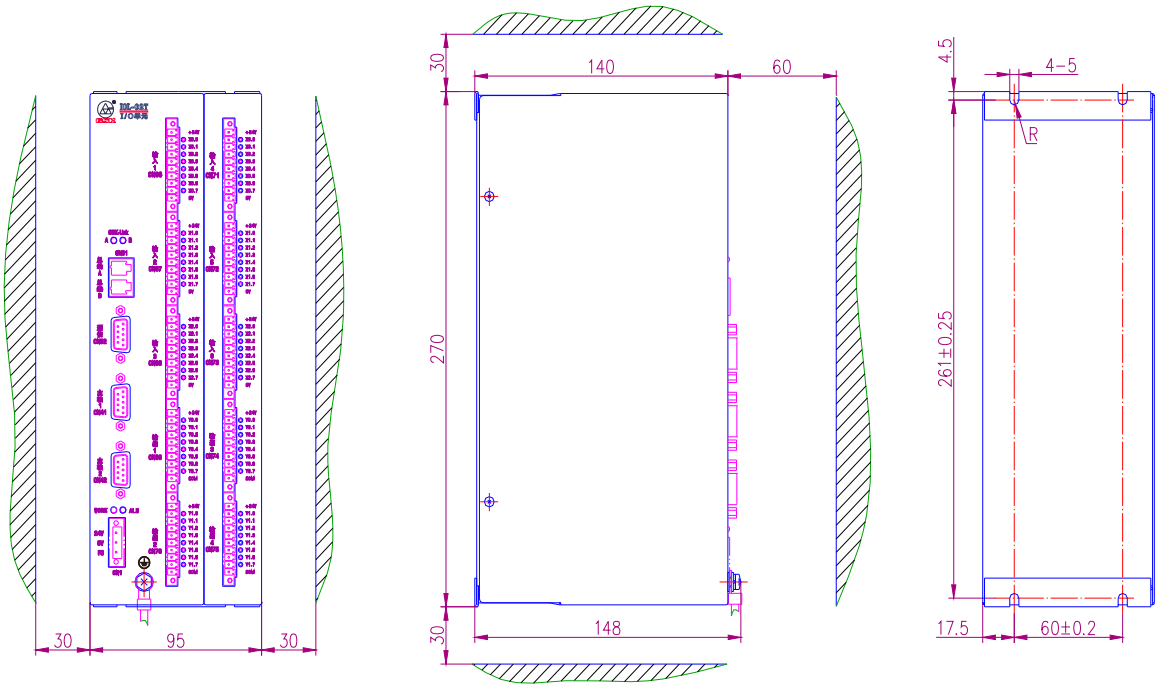


Fig.1-12

1.1.6.3 IOL-02F Appearance Dimension

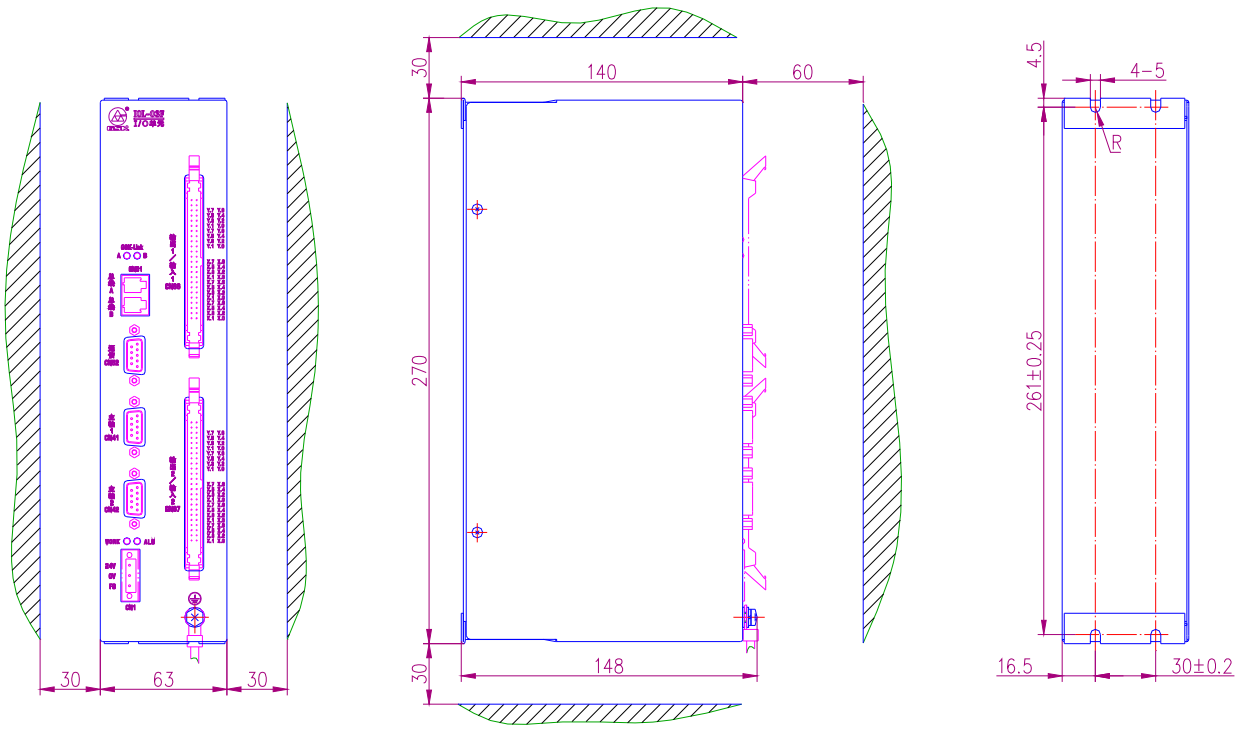


Fig.1-13

1.2 GSK988TA/988TA1/988TB Control System Constitution

1.2.1 CNC Rear Cover Interface Layout & Connection Figure

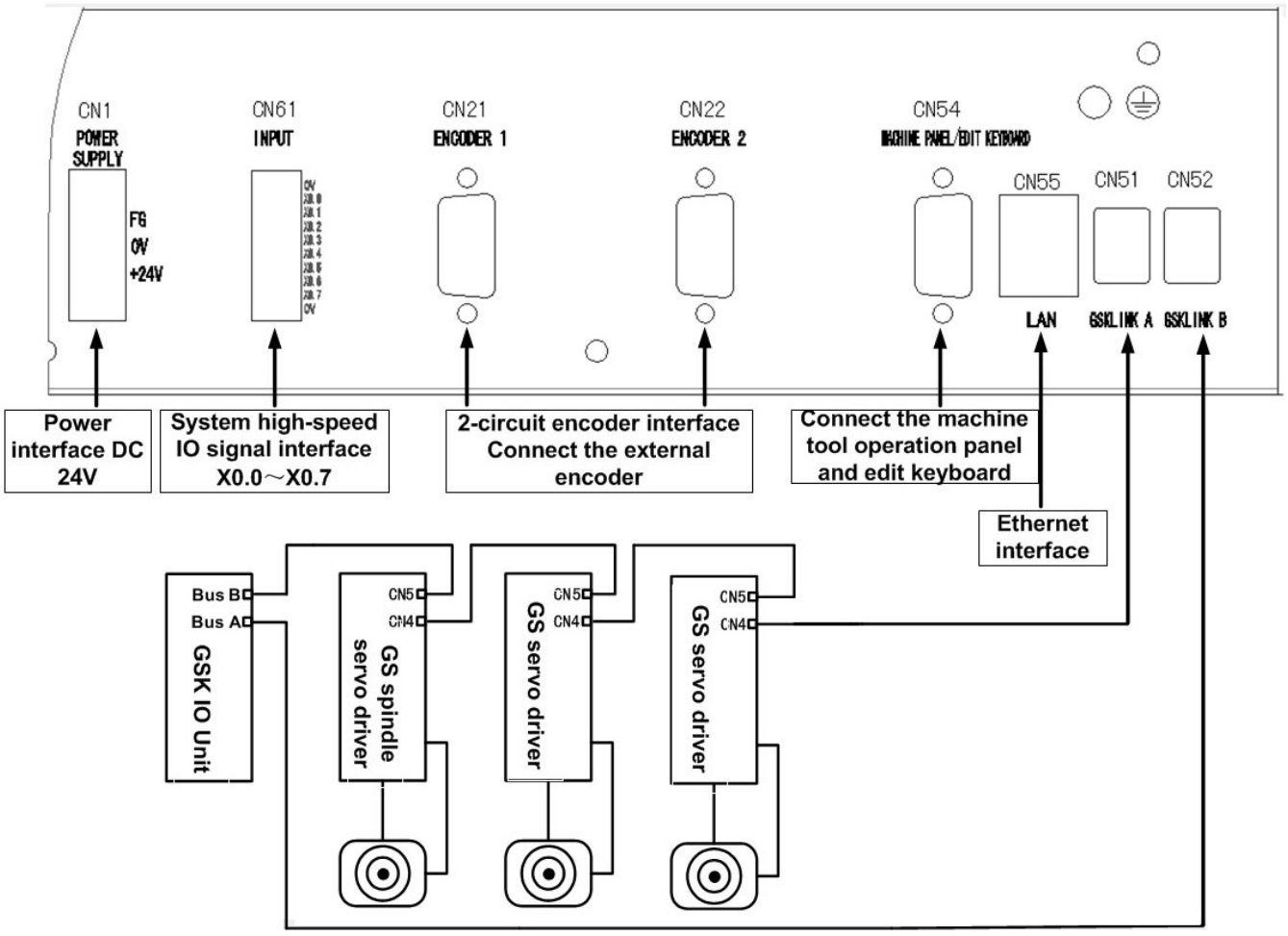


Fig.1-14

1.2.2 Rear Cover Interface of Machine Tool Operation Panel & Connection Figure

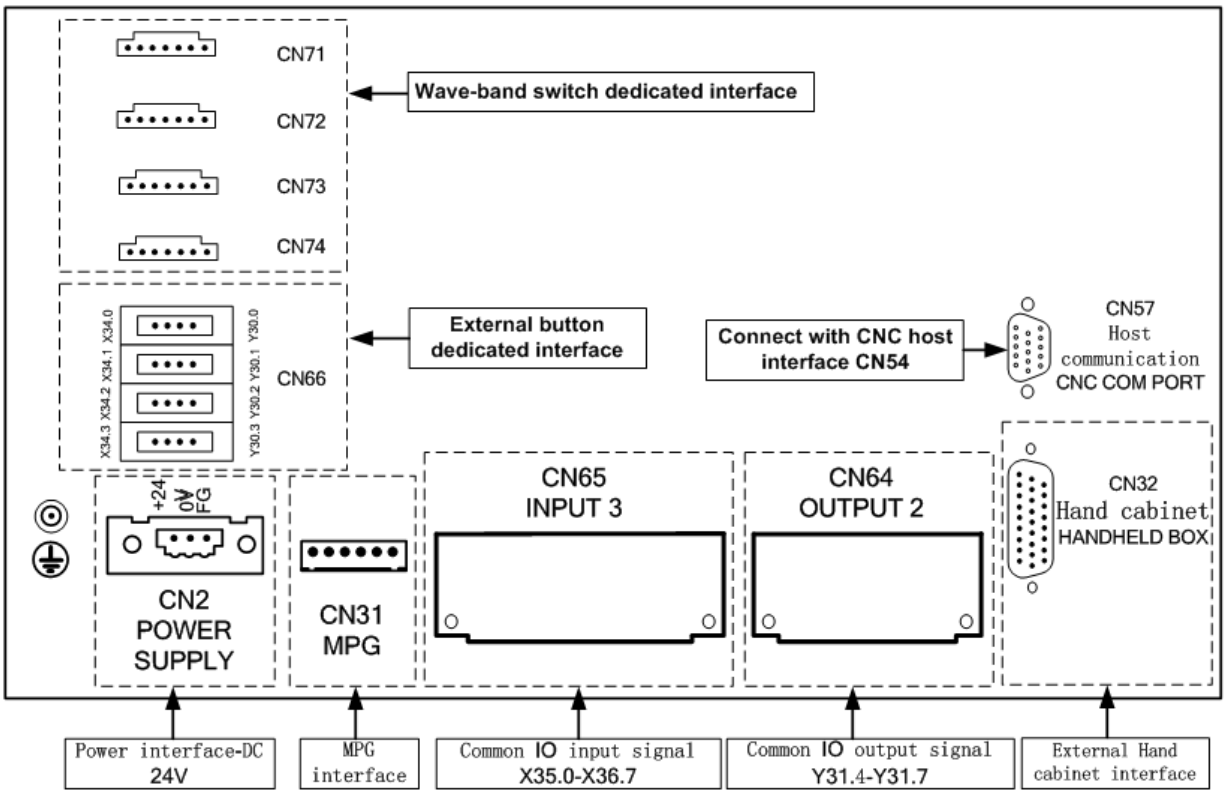


Fig.1-15

1.3 GSK988TA/988TA1/988TB Installation

1.3.1 Installation Condition of Electric Cabinet

- **The electric cabinet should be availably prevented the dust, coolant and the organic solution from entering it**
- **The distance between the CNC rear cover and case should be more than 20cm when designing the electric cabinet. It is necessary to consider that when the temperature inside the electric cabinet is upgraded, the temperature differences between inside and outside the cabinet should be less than 10℃.**
- **In order to guarantee the ventilation inside the cabinet, the fan can be installed in it.**
- **The display panel should be installed at the place where the coolant can not spray to it.**
- **It is important to consider reducing the electric interference from outside as much as possible when designing the electric cabinet to prevent the interference from delivering to CNC.**

1.3.2 System Grounding Requirment

The following grounding systems are provided to CNC machine tool:

- Signal grounding

The signal grounding offers the reference voltage (0V) from the telecommunication system;

- Frame grounding

The fram grounding is used on the safety, it is necessary to connect the shell of frame unit, panel and the shielding of the interface cable among units together. The frame grounding can be restrained the inside or ouside noise, too.

- System grounding

  The system grounding is used to connect the frame grounding of equipment and unit with the earth together.

1.3.3 Method of Anti-Interference

CNC is already adopted the anti-interference measures, such as the shielding space electromagnetism radiation, absorbtion impacting current, filtering power noise wave, etc. when designing, which can be prevented the external interference from impacting the CNC in some extent. In order to guarantee the stable operation of the CNC, the following measures should be performed when the CNC is being installed and connected:

① CNC should be depated from the interference equipment (For example, frequency-transformer, AC contactor, static generator, high-pressure generator and the section equipment of the dynamic circuit, etc.)

② Supply the CNC via the insulation transformer; the machine tool with CNC should be grounded; and the CNC and drive unit should be connected the separated grounding cable from the grounding point.

③ Barrage jamming: Connect the RC circuit (Refer to Fig. 1-16-1) in parallel at both AC coil ports; There is no alternation other than closing to the sensitive loading when RC circuit is installed; reversely perform the fly-wheel diode (Refer to the Fig. 1-16-2) in parallel both ports of the DC coil; Connect the surge absorber (Refer to the Fig. 1-16-3) in parallel at the winding port of AC motor

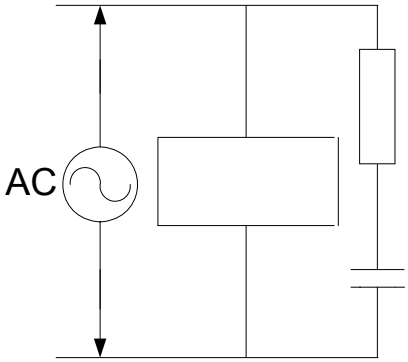


Fig. 1-16-1

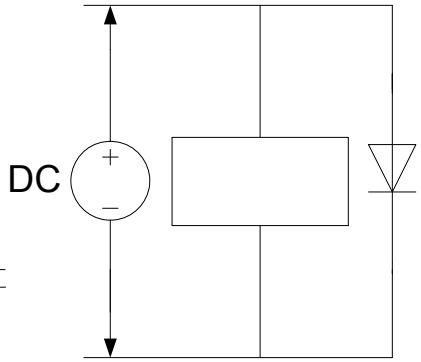


Fig. 1-16-2

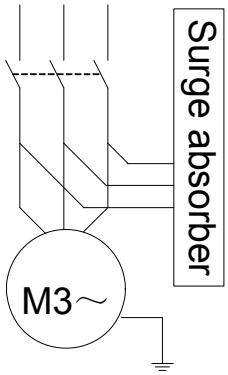


Fig. 1-16-3

④ The leading-out cable of the CNC utilizes twisted shielding cable or shielding cable, the shielding layer of the cable adopts the single-port grounding at the CNC side, the signal cable should be shortened as much as possible.

⑤ In order to reduce the interference among the CNC signal cables and the strong current cables each other, the following principles should be observed when wiring.

Group	Cable type	Group	Cable type
A	AC power cable	B	DC coil (24VDC)
	AC coil		DC relay (24VDC)
	AC contactor		The cable between the CNC and strong electric cabinet
C	The cable between the CNC and drive unit		The cable between the CNC and machine tool

Wiring requirements:

① Cable uses the double-twist cable

② The cable of group A is separately bound to group B or C, and its distance among them should be 10cm at least; or perform the electromagnetism shielding for group A cable

③ Separately bind the group C cables to A, and its distance between them should be 10cm at least; or shield the group C cables; the distance between the group C cables and B should be 10cm at least;

④ Separately bind the group B to A; or shield the gourp B cables; group B cables should be far away from C as possible.

Chapter 2 Interface Signal Definition & Connection

2.1 CNC Host Interface Defintion & Connection

The interfaces of GSK988TA/988TA1/988TB are divided into one for the CNC front panel and 8 for the back one.

2.1.1 GSKLink Bus Interface

The bus interfaces of GSK988TA/988TA1/988TB are CN51 and CN52 (GSKLinkA and GSKLinkB), the interfaces are possessed feed servo drive unit with GSKLink bus communication function, spindle drive unit and extension I/O unit communication connection.

GSKLink bus communication connection cable is shown below:

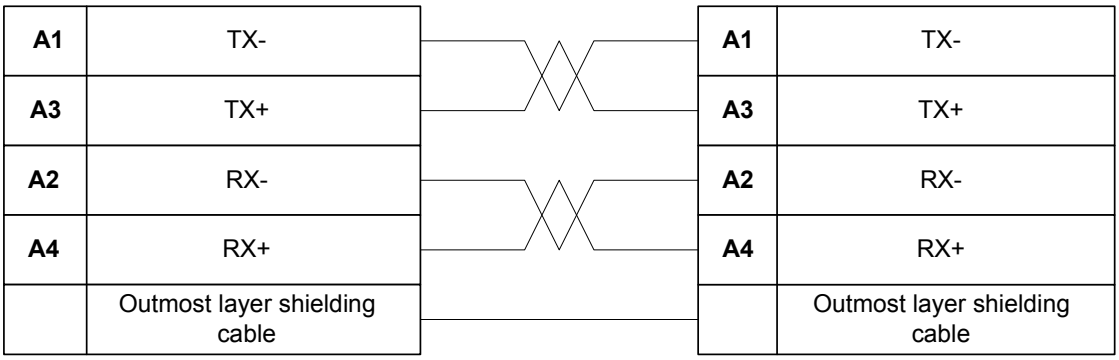
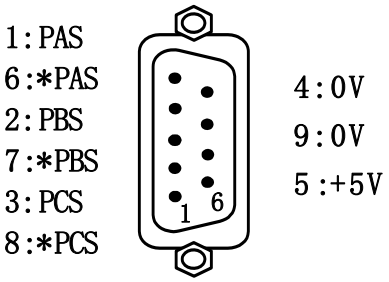


Fig.2-1 GSKLink communication connection

2.1.2 Spindle Encoder Interface

GSK988TA/988TA1/988TB owns two-circuit encoder input interface (N21, CN22), refer to the Fig. 2-2. Use the GSKLink interface to read the encoder spindle when using the GSKLink spindle instead of the spindle encoder.



Signal definition	Signal explation
*PAS/PAS	Encoder A phase pulse
*PBS/PBS	Encoder B phase pulse
*PCS/PCS	Encoder C phase pulse

Fig.2-2 CN21, CN22 encoder interface(9-core D-male socket)

Signal explanation

*PCS/PCS, *PBS/PBS and *PAS/PAS are differential input signals of the phase C, B and A of the

encoder separately; *PAS/PAS and *PBS/PBS are the orthogonal square-wave of the difference by 90°, the top signal frequency <1MHz; The encoder linear number use by GSK988TA is set by parameter No. 3720 (the linear number of the spindle encoder).

Spindle encoder interface connection

The connection between GSK988TA/988TA1/988TB and spindle encoder is shown in the Fig. 2-1-3; use the duble-twist cable when connecting (For example: ZLF-12-102.4BM-C05D encoder)

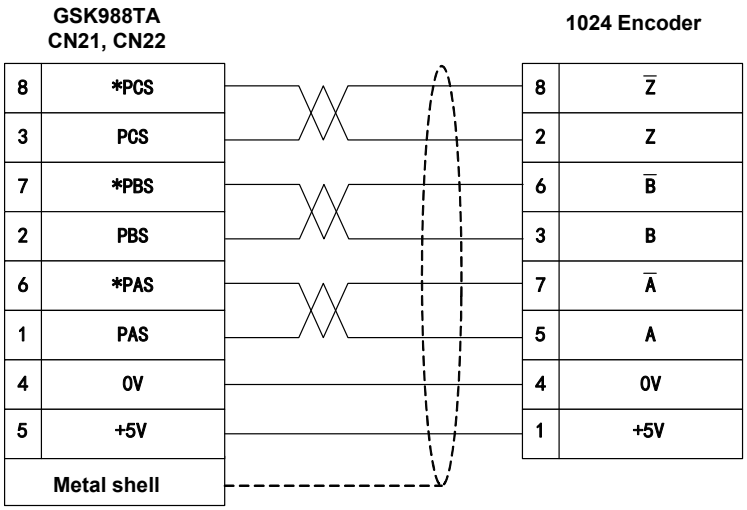


Fig.2-3 The connection between GSK988TA/988TA1/988TB and encoder

2.1.3 High Velocity Input Interface

GSK988TA/988TA1/988TB system equips with the high velocity I/O interface CN61 of 1 input signal, its address is X0.0~X0.7

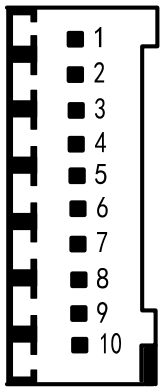
Input port CN61	CN61 pin No.	PLC add.
 CN61	1	GND
	2	X0.0
	3	X0.1
	4	X0.2
	5	X0.3
	6	X0.4
	7	X0.5
	8	X0.6
	9	X0.7
	10	GND

Table 2-1

2.1.4 Communication Interface

GSK988TA/988TA1/988TB system and machine operation panel are connected with the communication. Refer to the Fig. 2-1-4 for the interface pin

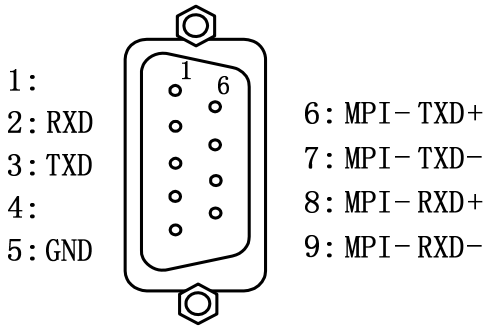


Fig.2-4

Signal	Explanation
RXD	The special-purpose communication signal of GSK988TB editing keyboard: Accept the data difference signal
TXD	The special-purpose communication signal of GSK988TB editing keyboard: Deliver the data difference signal
MPI-RXD+	Operation panel special-purpose communication signal: accept the data difference signal
MPI-RXD-	Operation panel special-purpose communication signal: accept the data difference signal
MPI-TXD+	Operation panel special-purpose communication signal: deliver the data difference signal
MPI-TXD-	Operation panel special-purpose communication signal: deliver the data difference signal
GND	Power 0V

Connect with the GSK988TA1 and GSK988TA machine operation panel

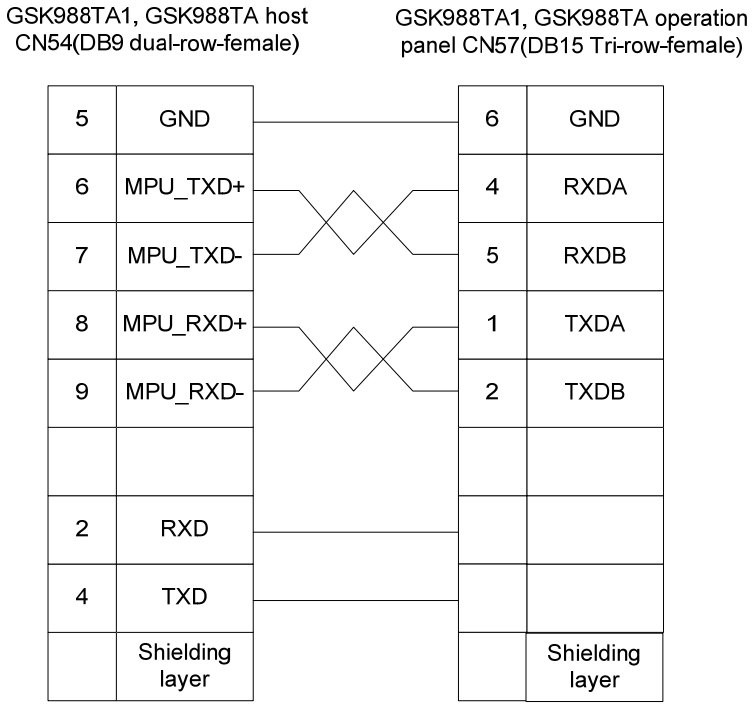


Fig.2-5

Connect with the GSK988TB machine operation panel and edit keyboard

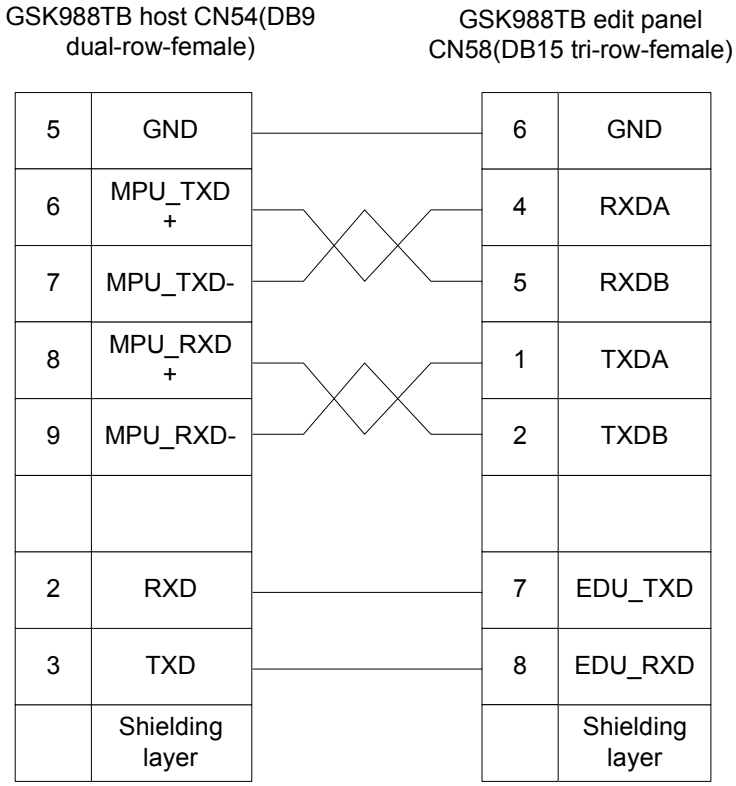


Fig.2-6

The connection between Edit panel and Operation panel of the GSK988TB

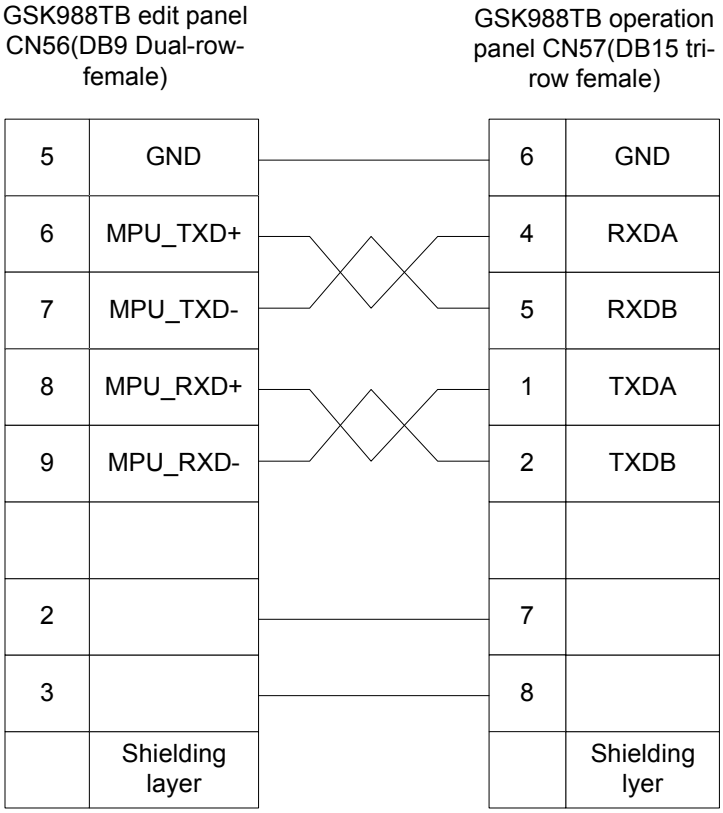


Fig. 2-7

Remark: The above-mentioned connections are used with a same cable, what the difference is the treatment of the internal equipment.

2.1.5 CNC Power Interface

GSK988TA/988TA1/988TB uses DC 24V power supply; the power interface definition is as Fig.2-8.

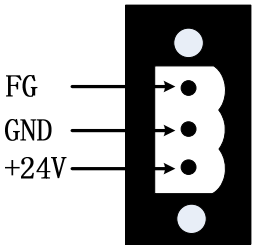


Fig.2-8 GSK988TA/988TA1/988TB system power interface CN1 pin definition

2.1.6 Network Interface

Network interface (Standard interface):

Pin No.	Signal	Pin No.	Signal
1	TXDLAN+	9	LINK_LED
2	TXDLAN-	11	LAN_LED
3	RXDLAN+	10、12	VDD33
6	RXDLAN-	13、14	Shell grounding

Table 2-2

Note: TXD+ and TXD- are the difference signal, so did RXD+ and RXD, which are connected with the dual twist cable connection

2.1.7 USB Interface of CNC Panel

USB interfaece (Standard):

Pin No.	Signal
1	VCC(+5V)
2	USB_DN0
3	USB_DP0
4	GND
5, 6	Shell grounding

Table 2-3

2.2 CNC Operation Panel

2.2.1 Dedicated Wave Band Switch Interface

The pin and address distribution of the dedicated wave band switch interface (CN71～CN74) is as the Fig. 2-9

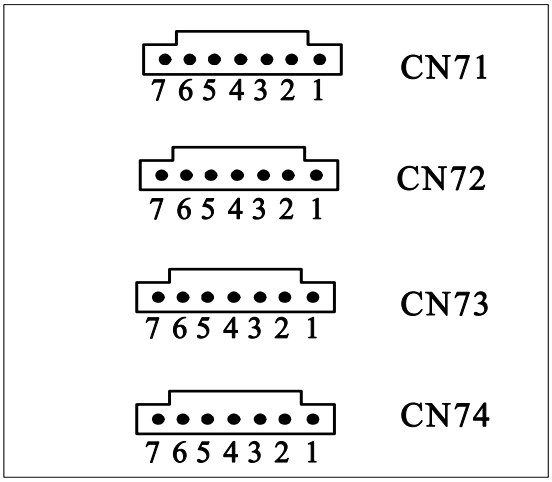


Fig.2-9

Interface	Pin No.	PLC add.	Interface	Pin No.	PLC add.
CN71	1	X30.0	CN73	1	X31.0
	2	X30.2		2	X31.2
	3			3	
	4	+5V		4	+5V
	5	X30.3		5	X31.3
	6	X30.1		6	X31.1
	7	0V		7	0V
CN72	1	X30.4	CN74	1	X31.4
	2	X30.6		2	X31.6
	3			3	
	4	+5V		4	+5V
	5	X30.7		5	X31.7
	6	X30.5		6	X31.5
	7	0V		7	0V

Table 2-4

Note: The dedicated interface of wave band switch is 5V input signal, only the wave band switch can be connected, and the system may be burnt if the connection is incorrect.

2.2.2 Dedicated Interface of The External Button

The pin and address distribution of the dedicated operation panel external button interface (CN66) is as Fig. 2-10.

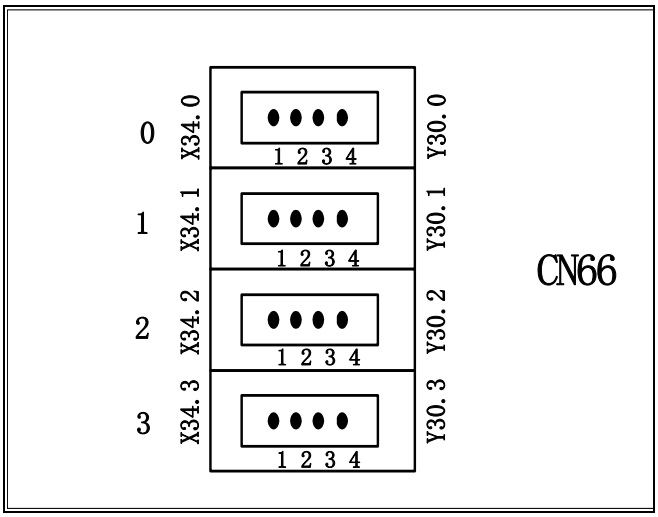


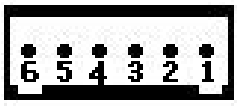
Fig. 2-10

Interface	Sequence No.	Pin No.	PLC add.
CN66	0	1	+24V
		2	+24V
		3	Y30.0
		4	X34.0
	1	1	+24V
		2	+24V
		3	Y30.1
		4	X34.1
	2	1	+24V
		2	+24V
		3	Y30.2
		4	X34.2
	3	1	+24V
		2	+24V
		3	Y30.3
		4	X34.3

Table 2-5

2.2.3 MPG Interface

MPG interface CN31 definition is as the Fig. 2-11



1：GND
2：HB+
3：HB-
4：HA+
5：HA-
6：VCC

Signal defintion	Signal explanation
HA+，HA-	MPG A phase signal input
HB+，HB-	MPG B phase signal input
GND	0V
VCC	+5V

Fig. 2-11 CN31MPG Interface

Hand cabinet interface CN32 definition is as Fig. 2-12

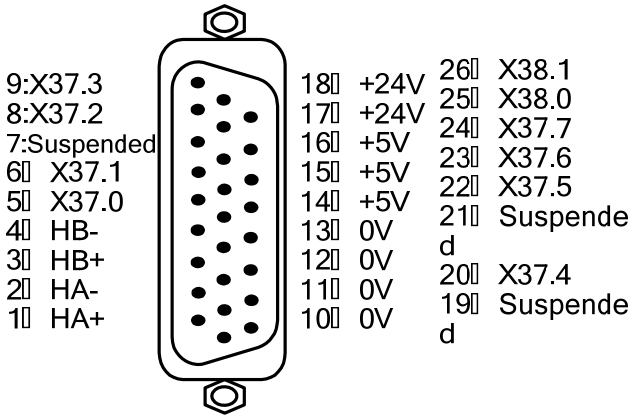


Fig. 2-12 CN32 MPG interface
 26-core type D pin socket

Signal defintion	**Signal explanation**
HA+, HA-	MPG A phase signal input
HB+, HB-	MPG B phase signal input
X37.0~X38.1	PLC signal address, switch value input

Signal explanation:

HA+, HA- and HB+, HB- are separately regarded as the differential input signal of phase A, B.

X37.0~X38.1 signals are the input addresses of PLC interface definition, which is used axle-selection of the external hand cabinet and the shift signal input.

X37.0~X38.1 signals are enabled with High Level input.

Connection with the MPG interface

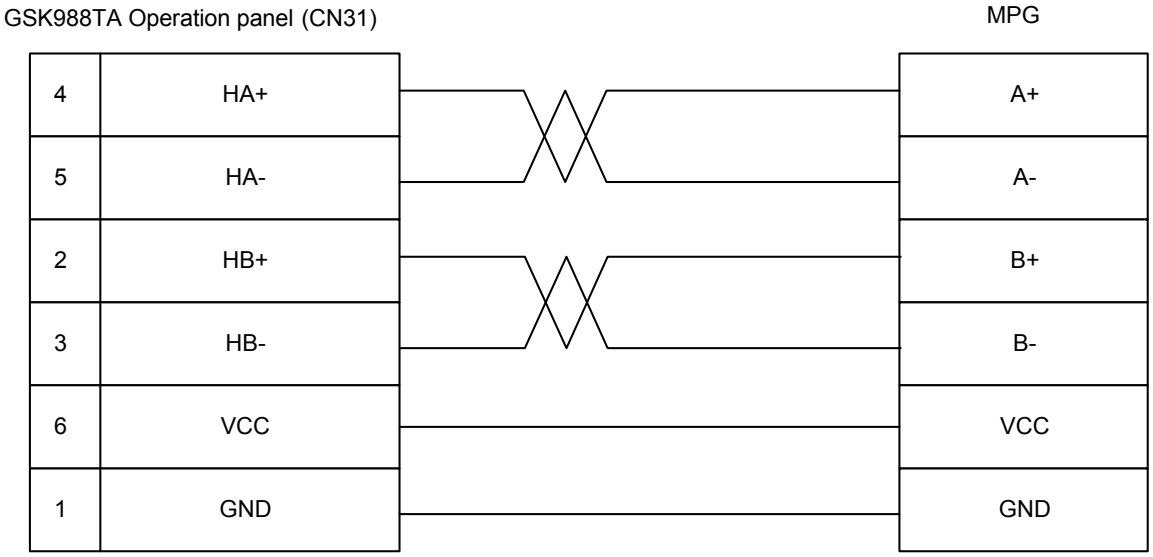


Fig.2-13 GSK988TA CN31 connection with MPG

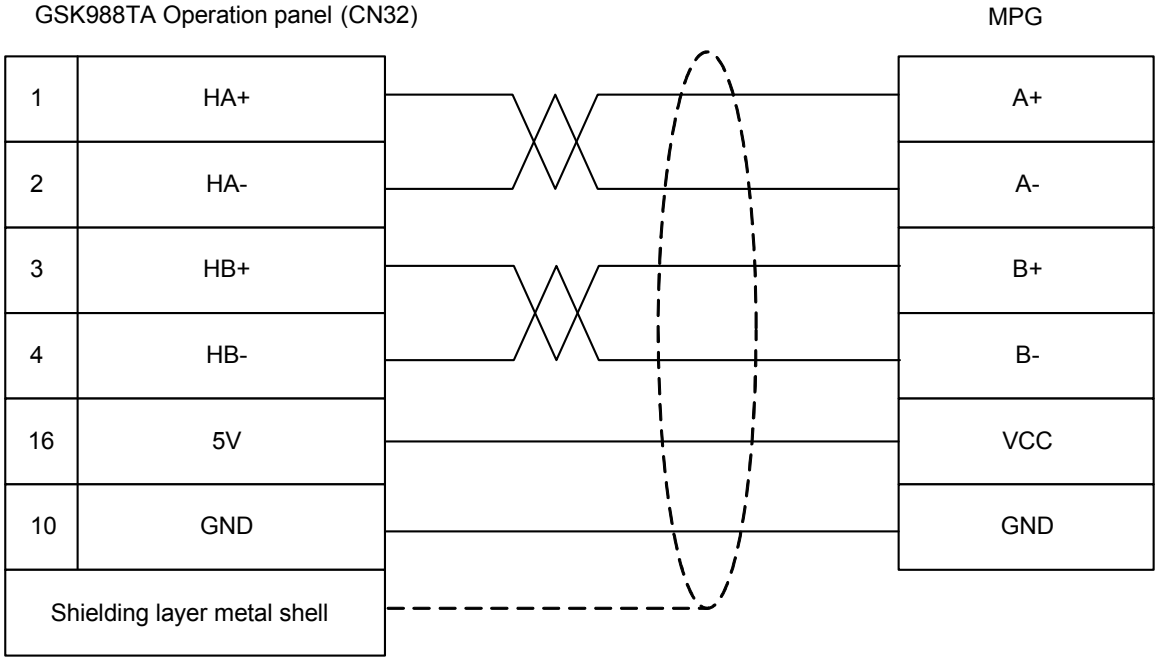


Fig.2-14 GSK988TA CN32 connection with MPG

Note: If it is connected the non-differential MPG, the HA- HB- signals are disconnected.

2.2.4 Communication Interface

GSK988TA/988TA1/988TB system and machine tool operation panel are adopted the communication connection method.

Refer to the Fig. 2-15 for the machine operation panel communication interface definition:

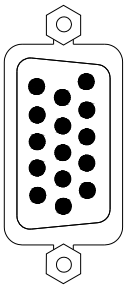


Fig.2-15 Standard machine tool operation panel interface CN57 (15-core D-type male socket)

Pin No.	Signal	IN/OUT	Explanation
1	RXDA	IN	Accept the data difference signal
2	RXDB	IN	Accept the data difference signal
4	TXDA	OUT	Deliver the data difference signal
5	TXDB	OUT	Deliver the data difference signal

Note: Refer to the Section 2.1.4 for the connection cable with CNC

2.2.5 Address Definition of Universal Input/Output

The universal input/output singal locates at the rear cover of the machine tool operation panel, and leads to the terminal for the reservation address to the user: 8 input points X36 and 4 output points Y31.

2.2.6 Power Interface Definition of Machine Tool Panel

Machine tool operation panel uses the DC24V, and its power interface definition is as the Fig. 2-16:

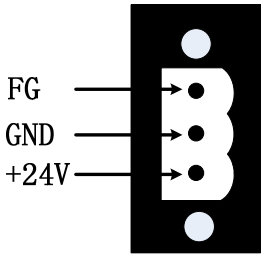


Fig.2-16 The CN2 pin definition of operation panel power interface

2.3 I/O Unit

2.3.1 Bus Interface CN51

CN51 bus A and B are used to connect with the CNC bus interface, its connection method is identical with the CNC GSKlink.

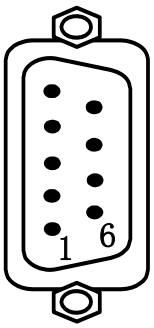
2.3.2 Communication Interface CN52

485 communication interface, reserved interface.

2.3.3 Spindle CN41 and Spindle CN42

The extension IO unit contains 4 circuits analog voltage output interface, which distributes on the CN41 and CN42 interfaces to connect the non-bus spindle with the analog voltage output; however, the bus spindle needs not connect it, the interface definition is as follows.

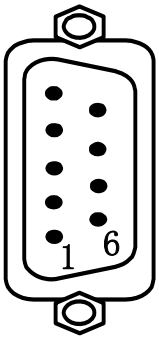
1:GND
2:IO-AOT0
4:GND
5:IO-AOT1



The 1st spindle CN41 interface

Signal definition	Signal explanation
1：GND	The 1st circuit analog voltage output grounding
2：IO-AOT0	0~+10V the 1st circuit analog voltage output
4：GND	The 2nd circuit analog voltage output grounding
5：IO-AOT1	0~+10V the 2nd circuit analog voltage output

1:GND
2:IO-AOT2
4:GND
5:IO-AOT3



The 2nd spindle CN42 interface

Fig.2-17

Signal definition	Signal explanation
1：GND	The 3rd circuit analog voltage output grounding
2：IO-AOT2	0~+10V the 3rd circuit analog voltage output
4：GND	The 4th circuit analog voltage output grounding
5：IO-AOT3	0~+10V the 4th circuit analog voltage output

2.3.4 Power Interface CN1

The extension I/O unit adopts DC 24V, its power interface definition is as follows.

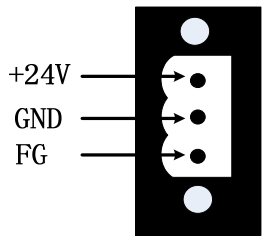


Fig.2-18

2.4 Usage of Input/Output Signal of CNC

The input/output signal of CNC contains: High velocity input/output signal on the host, wave-band switch dedicated signal on the machine tool operation panel, and the external button dedicated signal and current input/output output signal both on the operation panel.

2.4.1 Signal Input

The signal input is from the machine tool electric circuit or machine panel to CNC, when its interface input point is switched on with +24V (Note: The dedicated interface of wave brand switch is switched on with +5V), the input is enabled, the corresponding X address signal state is 1; when its interface input point is switched off with +24V, the input is disabled, the corresponding X address signal state is 0. The contactor both the machine tool sides for the input signal should be met the following conditions:

Contactor capacity: DC30V, above 16mA

The leakage current between contactors in the open-circuit: Below 1mA

The voltage/potential drop between contactors in the close-circuit: Below 2V (Current is 8.5mA, and the potential drop of the cable is included)。

There are two methods of external signal input: one is the contactor switch input, refer to the Fig. 2-19.

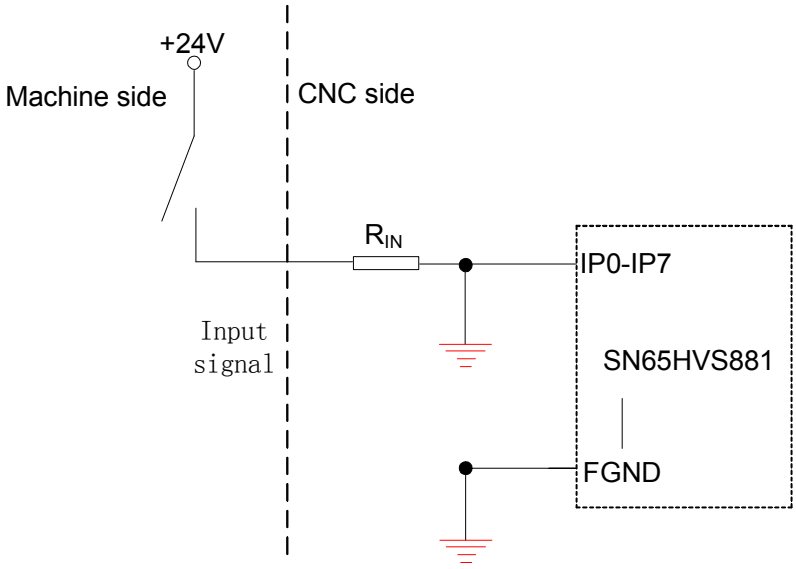


Fig. 2-19 Contactor switch input

The other one is without contactor switch (transistor) input, refer to the Fig. 2-20, 2-21.

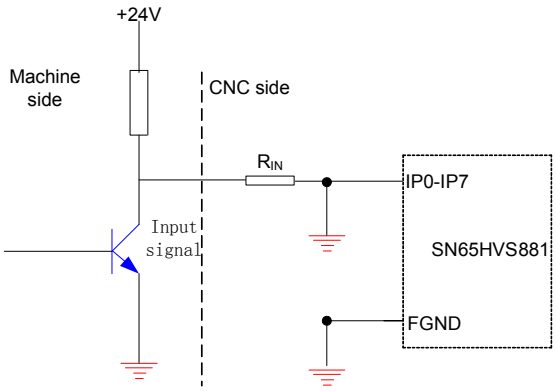


Fig. 2-20 NPN type connection

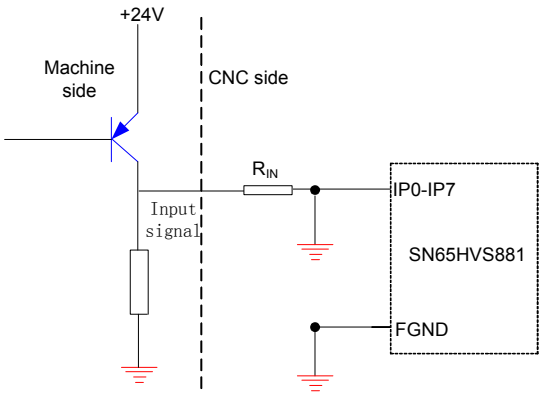


Fig. 2-21 PNP type connection

2.4.2 Signal Output

Output signal is used to drive the relay and indicator at the electric circuit side or machine panel side, when the output is enabled, the corresponding Y address output state is 1, and this output interface potential is 0V; when the output is disabled, the corresponding Y address output state is 0, and this output interface is high resistance state. The electric circuit is shown as Fig. 2-22:

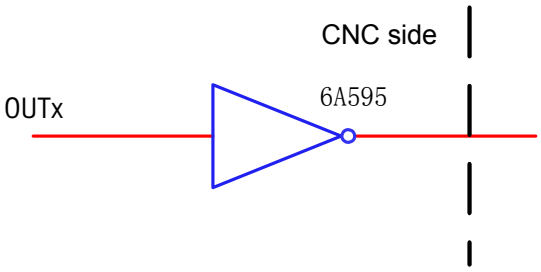


Fig. 2-22 The circuit structure of internal signal output

Therefore, there are two output states of output signal: 0V output or high-impedance. The typical

application is as follows.

➢ Drive the LED

It is necessary to connect a resistance in parallel to drive the LED, limit the current from the LED (Generally, it is 10mA or so). Refer to the Fig. 2-23：

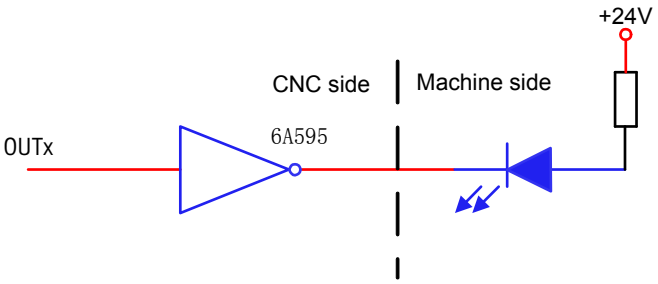


Fig.2-23

➢ Drive the filament type indicator

The filament indicator of drive out should be connected a preheating resistance to reduce the conduction impacting, the range of preheating resistance value is set based upon that the indicator is on the OFF state, refer to the Fig. 2-24:

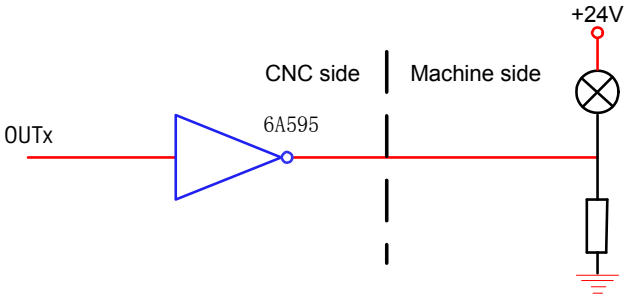


Fig.2-24

➢ Drive the induction loading (For example, relay)

The sensitive loading of drive output, in this case, it is necessary to connect the fly-wheel diode nearby the coil to protect the circuit output, and reduce the interference. Refer to the Fig. 2-25:

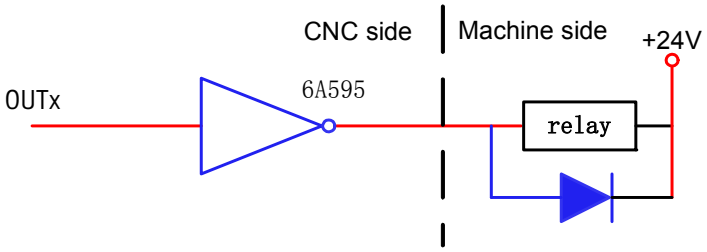


Fig.2-25

2.5 Usage of IO Unit Input/Output Signal

Different IO unit types with different configurations, the configurations of each type are shown below:

Type	Signal point input	Signal point output	Analog value input	Analog value output	Level input	Level output
IOL-01T	24 points	16 points		4-circuit	High	Low
IOL-02T	48 points	32 points		4-circuit	High	Low
IOL-02F	48 points	32 points		4-circuit	High	High

Note: The output signal of IOL-02F in IO unit is different with other types, which is High level output.

2.5.1 Signal Input

Refer to the Section 2.4.1 for the detailed usage.

2.5.2 Signal Output

2.5.2.1 Output Signal is Low Level

Refer to the Section 2.4.2 for the detailed usage.

2.5.2.2 Output Signal is High Level

The signal output is used to drive the relay and indicator at the machine electric circuit side or the machine tool panel, when the output is enabled, the corresponding Y address output state is 1, this output interface potential is +24V; when the output is disabled, the corresponding Y address output state is 0, this output interface is then regarded as High resistance. The electric circuit is shown as the Fig. 2-26:

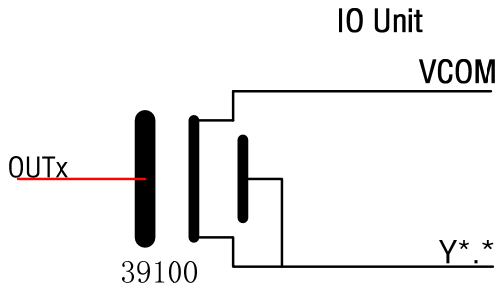


Fig. 2-26 The internal circuit structure diagram of output signal

Therefore, there are two output states of signal output: +24V output or high resistance. The typical applications are shown below:

- Drive LED

To output a drive LED needs serially connected

It is necessary to connect a resistance in series when outputting a drive LED for limiting the current flowed from the LED (Generally, it is about 10mA). Refer to the Fig.2-27:

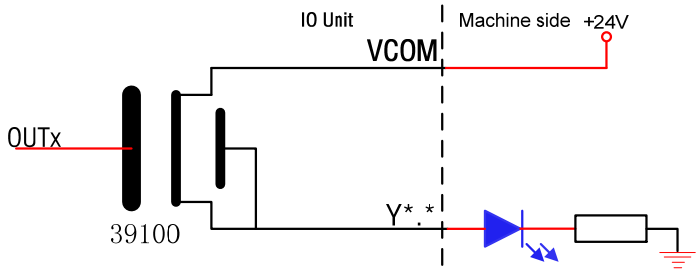


Fig. 2-27

➢ Drive filament indicator

The output drive filament indicator should be connected a preheating resistance from outside to reduce the current impacting from conducting; the preheating resistance value should be set based upon that the indicator is OFF, refer to the Fig. 2-28:

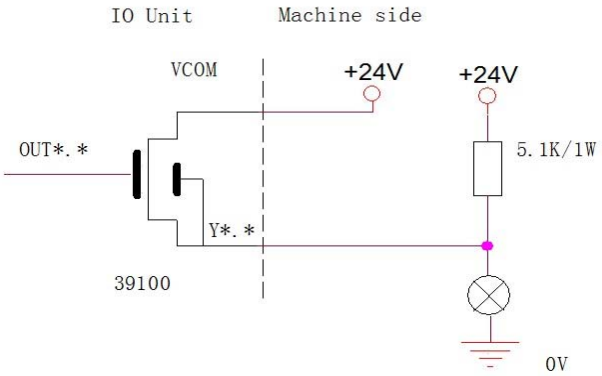


Fig. 2-28

➢ Drive sensitive loading (such as the relay)

Output the drive sensitive loading, in this case, it is necessary to connect the fly-wheel diode nearby the coil to protect the output circuit and reduce the interference. Refer to the following Fig. 2-29.

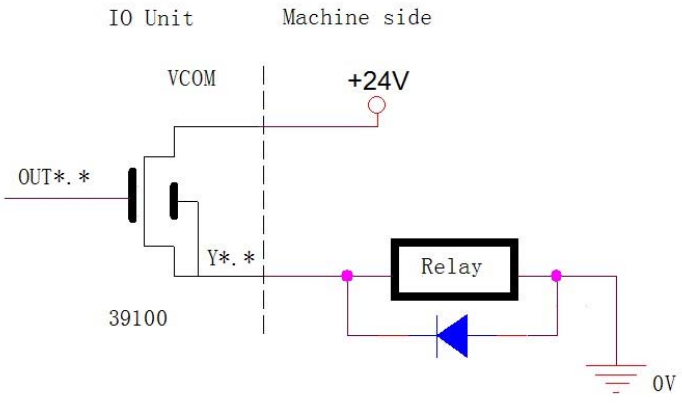


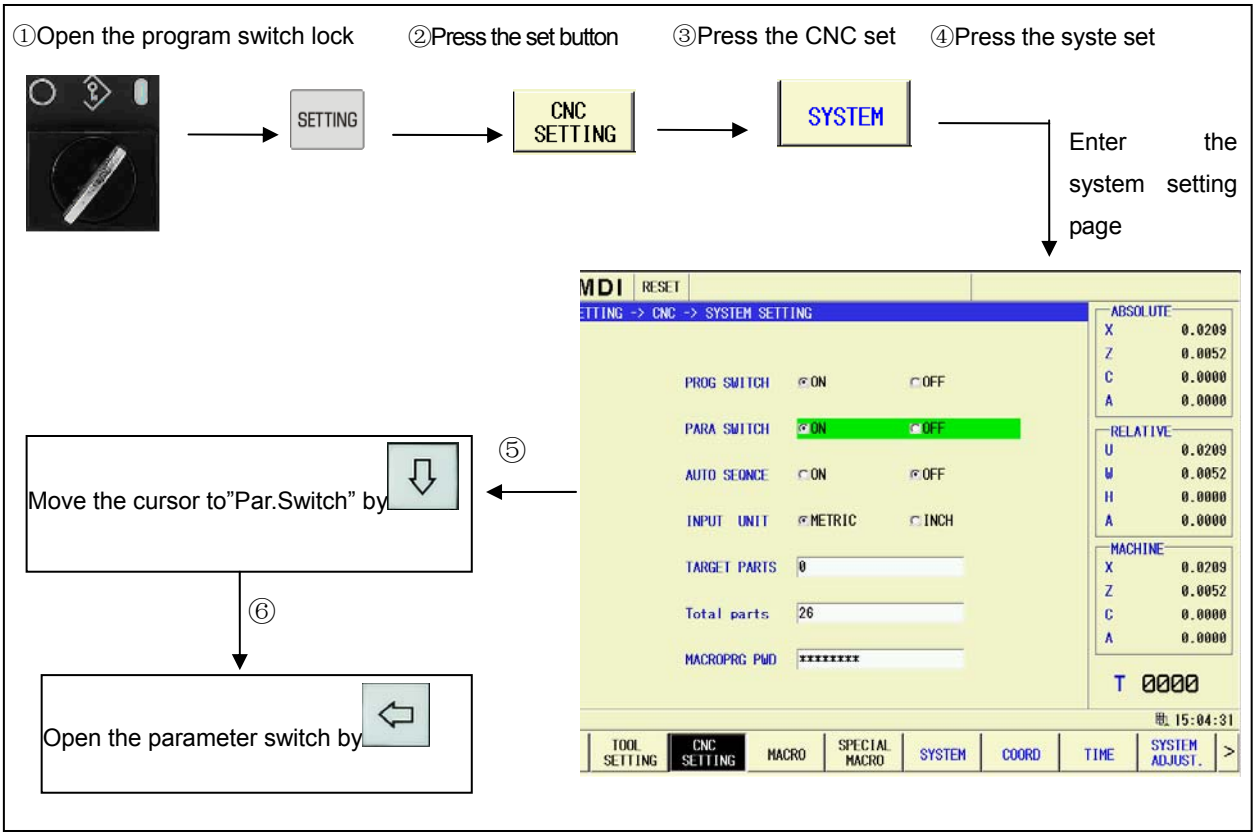
Fig. 2-29

BOOK II DEBUGGING

Chapter 1 Machine Tool Debugging & Operation

1.1 Parameter Setting

The modification, backup and recovery of the GSK988TA/988TA1/988TB system and servo parameters should be set above the equipment administration level (level 3); the parameter switch is at the ON state, it can be performed in the MDI mode. The operations of opening the parameter switch are shown below.



Note 1: After the system parameters are modified, some parameters can be immediately enabled, some can not unitll the system is turned on again. Refer to the Chapter Three: Parameter Explanation for details。

Note 2: If you want to check and modify the parameter for the servo in the CNC, the servo system should be correctly connection, as well the correct configuration of the servo slavery number.。

1.1.1 System Parameter

Successively: SYSTEM —> PARAM, then enter the system parameter setting interface.

In this page, either set the system parameter, or backup the parameter set by user at present, or

recover the parameter as the system default one or as the one of the user backup.

Note: Select calling the corresponding parameter based upon the configuration of this machine tool in the system parameter configuration page before the machine tool is being debugged

(1) The calling of parameter configuration

Under the 2-level authority, page to the extension softkey interface by 

in the system parameter page, then enter the parameter configuration list by 

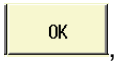
. Select the corresponding default parameter by 
, based upon the configuration of each axis for this machine tool, then the calling by 

, restart the system after selecting.

(2) Parameter search

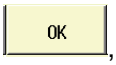
Method 1: Select the parameter to be checked or modified by 
, 
, 
or 
.

Method 2: Select different classification names by 

softkey based upon the parameter assortment search, then press the 

, the cursor is then positioned at the 1st one of this kind of parameter.

Method 3: Input the parameter No. to be selected by 

softky based upon the parameter number search, then press the 

, the cursor is then positioned at this parameter situation.

(3) The setting of bit-parameter

Method 1:

① After searching and positioning the parameter to be modified, press 

, so that the selected parameter is on the modifiable state.

② Input the 8-digit binary numerical value to be modified by the corresponding buttons, then complete the setting by 

(The “0” should be supplied at High-order when the inputted value is less than 8 digits).

③ Set the other parameters by 
, 
, 
and 
.

Method 2:

① After searching and positioning the parameter to be modified, select the parameter bit to be modified by  and  .

② Repeated press the  , so that the parameter bit can be shifted between 0 and 1, and then modify the value of this parameter bit.

③ Complete the setting by moving the cursor.

④ Set the other parameters by  ,  ,  and  .

(4) The setting of the numerical value parameter

① After searching and positioning the parameter to be modified, press  , so that the selected parameter is on the modifiable state.

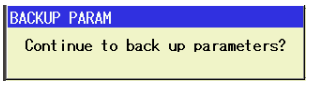
② Input the desired numerical value by its corresponding buttons, and then confirm the setting is performed by  .

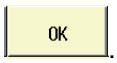
③ Set the other parameters by  ,  ,  and  .

(5) The backup and recovery of parameter

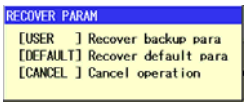
Firstly, user can perform the parameter backup by  before altering the parameters. When the parameter modification is incorrect or ignored, press the  softkey, the parameter is then recovered as the original one or as the system default one.

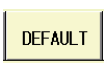
Parameter backup:

① Press the  softkey below the parameter page, the  displays.

② The parameter backup set at present by user can be performed by  .

Parameter recovery:

①  displays by the  softkey.

② Recover the parameter to the backup one from the user by  softkey; recover the parameter to the system default one by  ; and then retreat from the recovery

parameter interface by CANCEL.

1.1.2 Servo Parameter

(1) **The modification and save of servo parameter**

After the GSKLinK communication is normal, enter the servo parameter interface by pressing the GSKLink → SERVO → SERVO PARAM in the system page set.

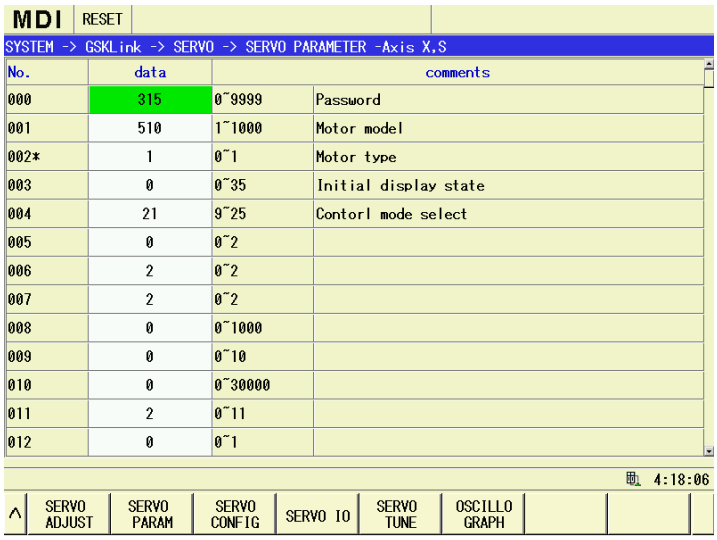


Fig. 1-1

Servo parameter page can be searched, modified, saved, and backup and recovered the servo parameter, as well recover the motor's default parametr and the leading servo parameter.

① **Axis parameter pamage shift:** Shift among the X axis, Z axis and S axis by Axis X, Axis Z and Axis S, and then display the servo parameter from the corresponding axis.

② **Parameter modification:** Move the cursor to the parameter position to be modified, after inputting the parameter value, and then press the INPUT to complete it.

③ **Parameter save:** Write the parameter to servo by SAVE after the servo parameter is modified; the altered parameter value remains unvariable after the servo is turned on again.

④ **Parameter backup:** Directly perform the parameter backup to the stored save area by BACKUP.

⑤ **Parameter recovery:** Directly recover the parameter where from the backup of the saved area to the current one by RECOVER.

Note: CNC directly read the servo parameter after the power is turned on again; that is: the current displayed parameter in the CNC servo page is the one of the servo terminal.

(2) Recover the motor's default parameter

① Press the [LOAD DEF.PAR.] softkey on the servo configuration page.

② Select the current used motor type in the following sprung dialogue frame; The system may call the corresponding motor default parameter and then cover the current saved one as long as pressing the [OK] softkey.

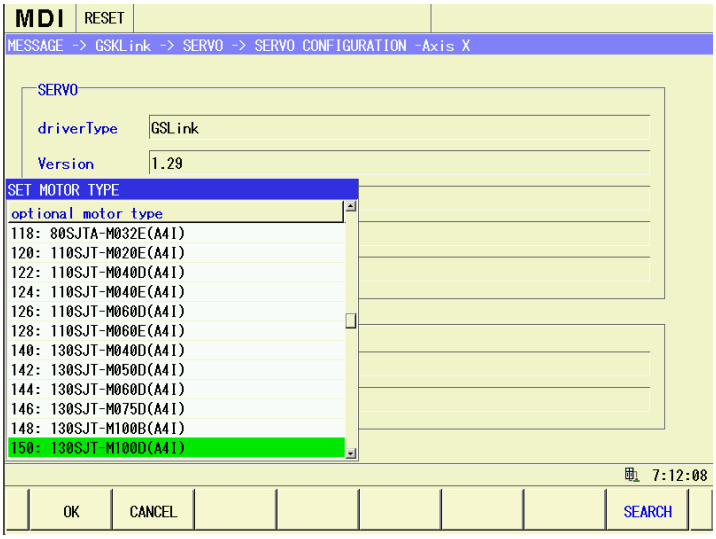


Fig.1-3

1.1.3 I/O Unit Parameter

(1) I/O unit configuration

After the GSKLink communication parameter setting is correct and the bus connection is normal, if it is firstly connected with the I/O unit, the system may alarm for prompting that the Ethernet communication does not establish due to the logic ID of the I/O unit is regarded as 0 by default.

Configure the logic ID of I/O unit by hand, firstly, modify the parameter №3050 to set the desired connected I/O unit numbers, and then modify parameters №3051～№3054; Modify the desired configured logid ID number within the parameter range, restart it after the power is turned off, and then press the [GSKLink]→[COMMUNICATION]; modifying the slave number is indentical with the №3051～№3054, and then it can be performed after restarting the I/O unit and CNC.

（2） I/O unit parameter setting

After the GSKLink communication is normal, press [GSKLink] → [>] → [I/O unit] →

I/O PARAM softkey to enter the I/O unit parameter interface in the system page set. If the system is connected IO unit at the first time, the parameter of I/O unit does not set, and the default PLC addresses are automatically distributed to DI and DO contactors from X100 and Y100; refer to the following figure:

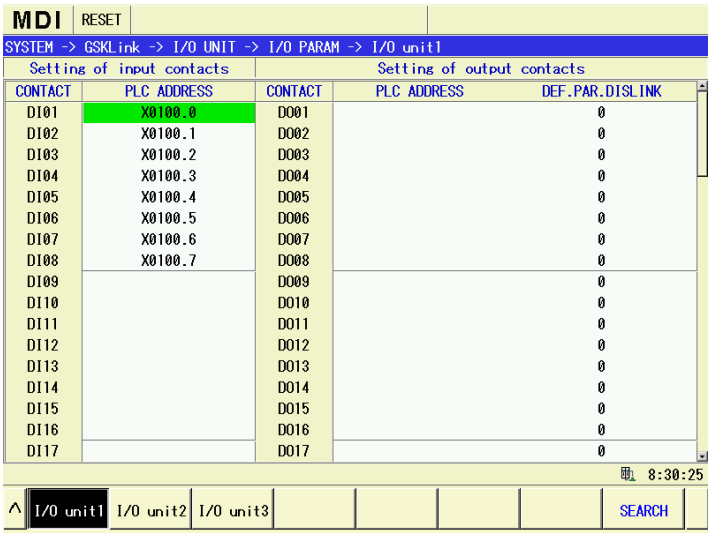


Fig. 1-4

The setting of input/output address: This address should be configured by hand. Move the cursor to the I/O type where the blank position at the right of DI, press the MODIFY, and then input 100; refer to the Fig. 1-1-1, press the OK, and the other steps of parameter are same: move the cursor to the desired setting position, and repeat the previous steps. **The port type of output address is the parameter of DO, and the setting method is consistent with the DI.**

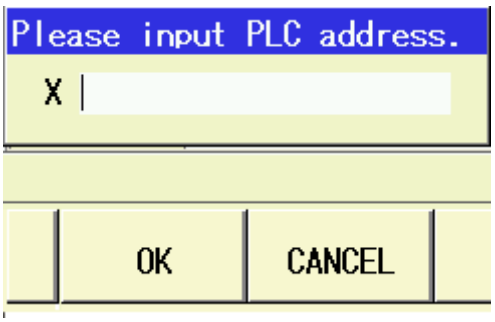


Fig. 1-5

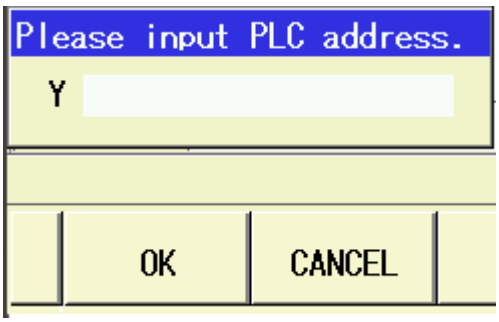


Fig. 1-6

Note: The standard ladder diagram of the GSK988TA/988TA1/988TB is already defined the I/O unit interface address. It is necessary to configure the I/O unit based upon the defined address of the ladder diagram if the system built-in standard ladder diagram is used; refer to the following table:

I/O unit interface	Add. definition
CN66 input 1	X100.0～X100.7
CN67 input 2	X101.0～X101.7

CN68 input 3	X102.0～X102.7
CN71 input 4	X103.0～X103.7
CN72 input 5	X104.0～X104.7
CN73 input 6	X105.0～X105.7
CN69 input 1	Y100.0～Y100.7
CN70 input 2	Y101.0～Y107.7
CN74 input 3	Y102.0～Y102.7
CN75 input 4	Y103.0～Y103.7

Table 1-1

1.2 Usage of U Disk

The U disk function of the GSK988TA/988TA1/988TB supports the bidirectional transmission of the machining procedure, PLC program, parameter, cutter and screw compensations files, etc. The U disk can be operated in the three pages, such as the file administration, program and ladder diagram.

1.2.1 File Administration Page

When the U disk in the system USB port, press the MEMORY DEVICE entering the file adminstration page in the SYSTEM page:

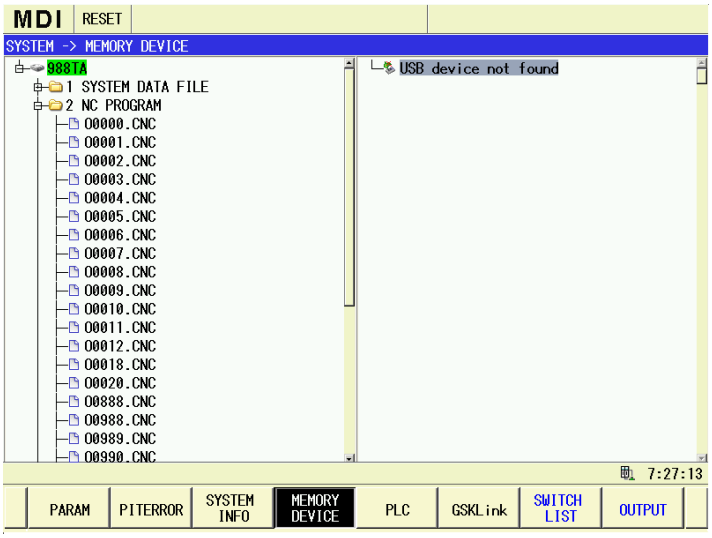


Fig. 1-7

This page can be performed the bidirectonal transmission (CNC－>U, U－>CNC) to the system files (system parameter, cutter compensation and screw compensation, etc), ladder diagram file and component program. Refer to the following operations:

① Repeatedly shift the cursor between the system list and U disk list by SWITCH LIST.

② Select the desired file or list to be copied by ⇧, ⇩, and then select it by INPUT (Repeated press this key can be regarded as the selection/cancellation shift).

③ The selected program is copied to the local list or the one in the U disk by OUTPUT.

1.2.2 Program Page

When the U disk in the system USB port, enter the program list (refer to the following left figure) by PROGRAM, the extension softkey is then displayed by >. In the USB page, the program in the U disk can be performed variable operations such as load, copy, paste, create, saved as, deletion, rename and search, etc.

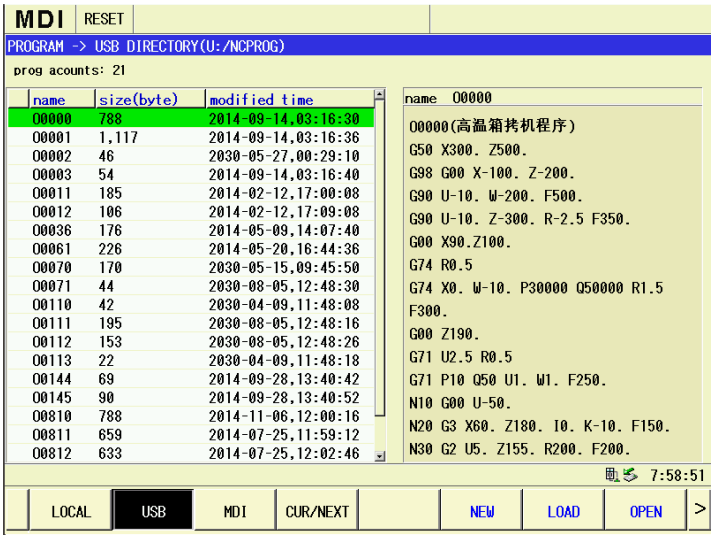
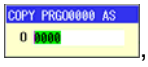


Fig. 1-8

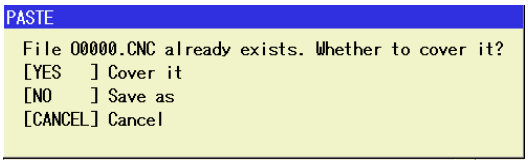
The bidirection transmission of program

① Both the system list and the U disk list can be shifted by LOCAL and USB.

② Move the cursor to the desired program to be copied by ⇧, ⇩, page to the extension softkey interface (refer to the above-mentioned right figure) by >, and the copy the selected program to the local list or the one of the U disk by OUTPUT softkey.

③ When the copied program is already existed, the displayed dialog prompts (refer to the left figure). Press the “YES” to save the existed program; “NO” springs out the dialog frame 

, it prompts that the inputted program name is saved as:, and then cancel the operation by “CANCEL”.



Note 1: Perform the U disk list in the program page → the machining program should be placed at the root-list "NCPROG" file of the U disk when the local list program is transfered.

Note 2: Perform the local list → the "NCPROG" file will be automatically created if there is no "NCPROG" folder when the U disk list maching program transfers.

1.2.3 Ladder Diagram Page

When the U disk in the system USB, press the PROGRAM in the ladder diagram page, then display the LOCAL interface; and then shift to the U disk list interface by USB.

Copy the program in the U disk list to the local list by OUTPUT softkey, vice versa.

The U disk list is regarded as an example, the concrect steps are shown below:

① Enter U disk file list by USB softkey;

② Select the desired ladder diagram program to be copied by ⇧ or ⇩, and then the selected program is copied to the local list by OUTPUT.

Note 1: Perform the U disk list in the ladder diagram page → the PLC program should be placed at the root-list "LDFILE" of the U disk when the local list PLC is transferred, and then the PLC program in the U disk can be read.

Note 2: Perform the local list ➜ When the "LDFILE" folder does not in the U disk, during the U disk list PLC transfers, and then the folder "LDFILE" is automatically created, and output the PLC program to this folder.

1.3 Operation of PLC

Enter the ladder diagram page set by SYSTEM function key and PLC softkey one after another, and this page is mainly consists of sub-page, such as the version information, monitoring, PLC data, PLC state and program list, etc., which can be viewed the following displayed content by its corresponding softkeys.

At the same time of entering the ladder diagram page set, the content of VERSION displays, refer

to the following figure. The VERSION page displays the information, such as the version information, the ladder digram program of current operation and its operation state, etc.

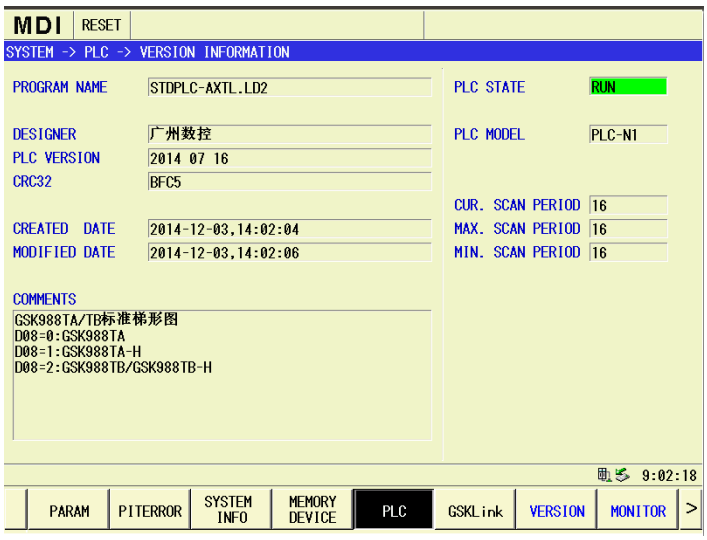


Fig. 1-9

1.3.1 Operation and Stop of PLC

In the ladder diagram page set, press the > softkey, and then press PROGRAM —> LOCAL step by step, the interface is as follows:

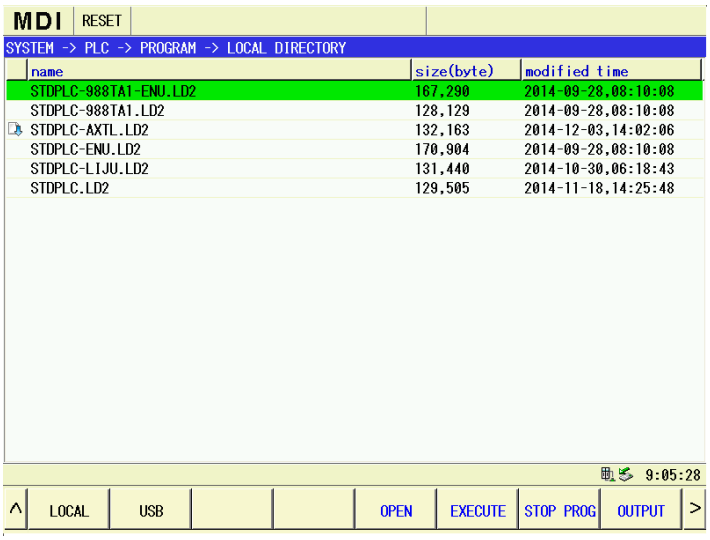


Fig. 1-10

In this page, select the PLC program by ⇧ or ⇩, and then open, perform and stop the PLC by softkey, as well output the program to U disk, then the program operation, such as the Save as, Creation, Deletion and backup gaining, etc., can be performed by > softkey.

The execution of PLC program

Select the PLC program by ⇧ or ⇩, and then operate the selected PLC program by EXECUTE.

Note: The PLC program of current operation is with the mark.

The stopping of PLC program

Move the cursor to the program being operated by ⇧ or ⇩, and then STOP PROG, the system will stay at the state without PLC operation; refer to the following interface:

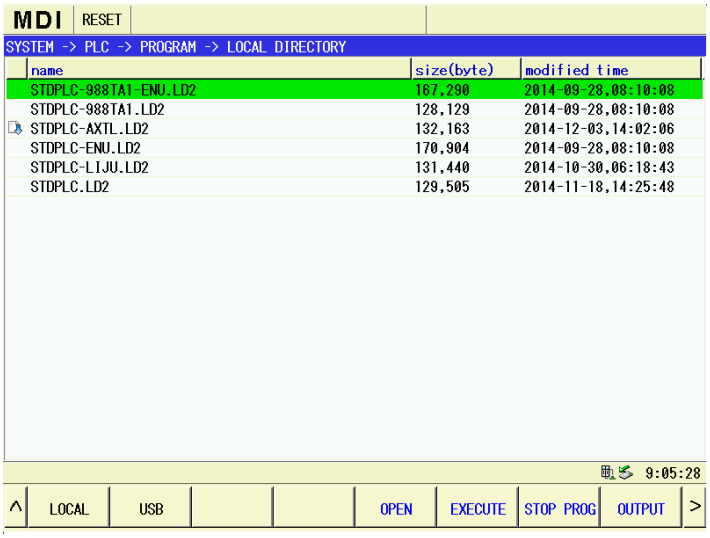


Fig. 1-11

1.3.2 Monitoring and Diagnosis of PLC

(1) The state monitoring of PLC program

Enter the operation monitoring display screen of the current operated ladder diagram program by MONITOR in the ladder diagram connection.

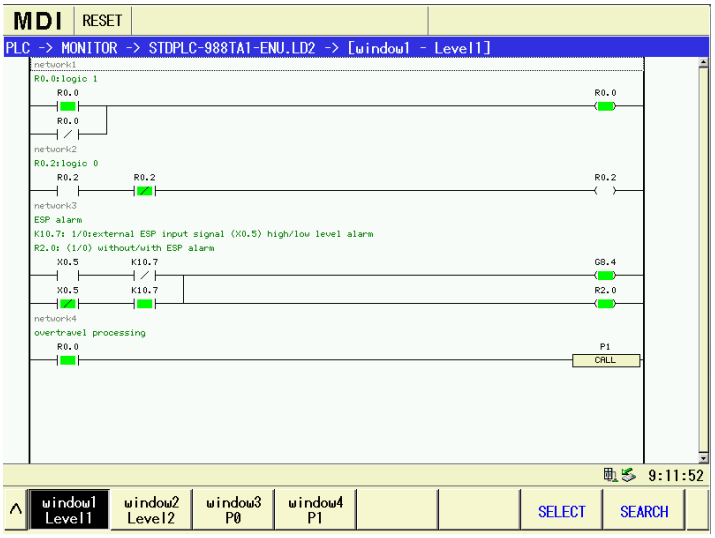


Fig. 1-12

The ON/OFF state of current contactor, coil and the current values of the timer and counter can be checked in the monitoring page. The green is regarded as the background when the contactor and coil are conducted; but if it is not conducted, the background is same as the window's color. For example, X0.5 ⊣■⊢ means that the contactor X0.5 conducts; Y25.2 ⊣( )⊢ means that the coil Y25.2 does not conduct.

The checking of window program

There are 4 blocks windows can be shifted and monitored at a rapid traverse rate in the monitoring page: [window1 Level1], [window2 Level2], [window3 P0] and [window4 P1]; The ladder diagram of corresponding block by its selected window is displayed in the screen by its related softkey of each window.

Note 1: Windows 1 to 4 are regarded as short-cut buttons, which can be checked the corresponding block in this window.

Note 2: The corresponding block of the windows 1 to 4 should be modified accordingly; it is not hold after the power is turned off. The previous 4 blocks in the corresponding ladder diagram is defaulted after the power is turned on.

The selection of the window block

① Select the window based upon its requirement.

② Press the [SELECT] softkey, the page is as follows:

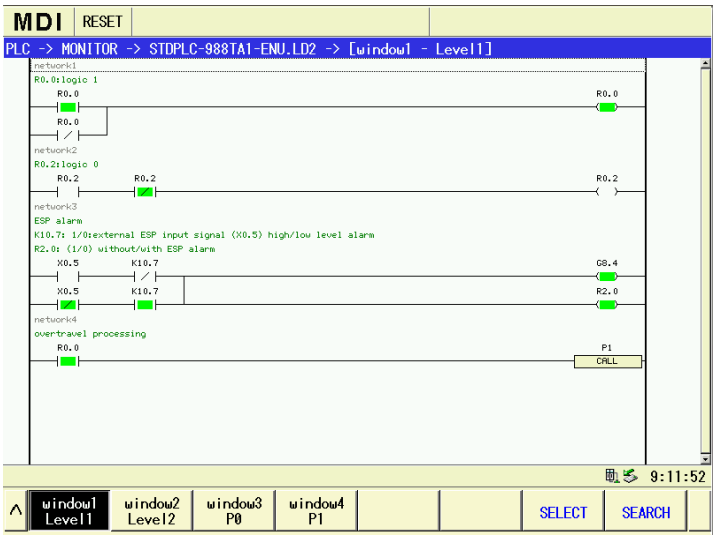


Fig. 1-13

③ The corresponding ladder diagram block for each window will be selected by [▤], [▥], [⇧] and [⇩].

④ Confirm and return the previous menu by [OK] softkey, and then cancel the selected operatoin and return to the previous menu by [CANCEL].

The searching of the parameter, command and network

① Select the block window to be searched the command, parameter and network, etc.; that is, select the window by controlling the [window1 Level1], [window2 Level2], [window3 P0] and [window4 P1] separately, so that its corresponding the program of block ladder diagram displays in the window, and then perform the searching of command, parameter and network, etc.

② Enter the searching page by [SEARCH], refer to the following figure:

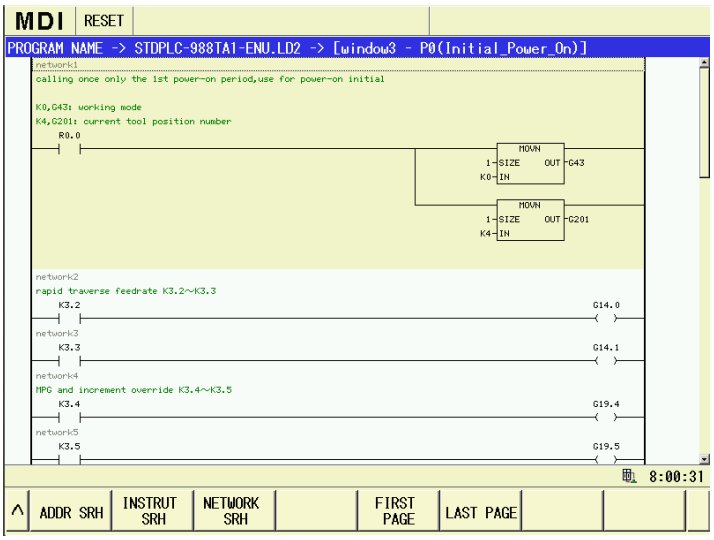


Fig. 1-14

③ Search the corresponding parameter, command and network in its window block by separately controlling the [ADDR SRH], [INSTRUT SRH] and [NETWORK SRH] softkeys, and the position the cursor to the corresponding position.

④ Position the cursor to the block's first line or the end line of its corresponding window to check by [FIRST PAGE] or [LAST PAGE].

(2) PLC state diagnosis

In the ladder diagram page set, enter to the PLC state display page by [>] and [PLC STATE] step by step; the display page is as the following left figure:

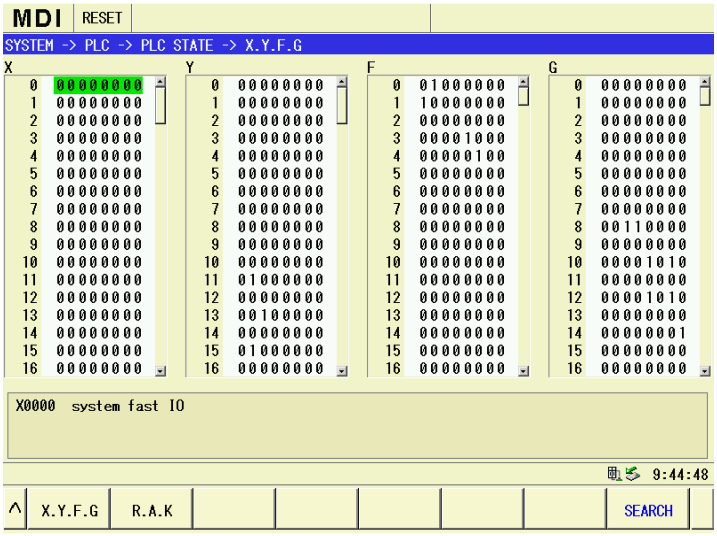


Fig. 1-15

Note: The note displayed in this diagnosis page is the PLC information of the current operation. The displayed information from diagnosis may differ from the variable PLC; the displayed note at present is defined by the compiler of the ladder diagram.

The checking of each signal state:

The state of X, Y, F and G signals are displayed in the window by [X.Y.F.G] softkey; and the R, A and K signal states are appeared in the window by pressing the [R.A.K] softkey.

The shifting can be performed between the signals X, Y, F and G or R, A and K by ⇦ or ⇨.

Checking can be performed in each signal of X, Y, F, G and R, A, K by , , ⇧ and ⇩.

Shift the in-position checking state by [CHANGE] softkey; and each state of signal position can be then checked.

1.3.3 Checking and Setting of PLC Data

Enter to the PLC data state display page by [PLC DATA] in the ladder diagram page set; the setting and saving of the K, D, DT and DC parameters can be performed. Refer to the display page:

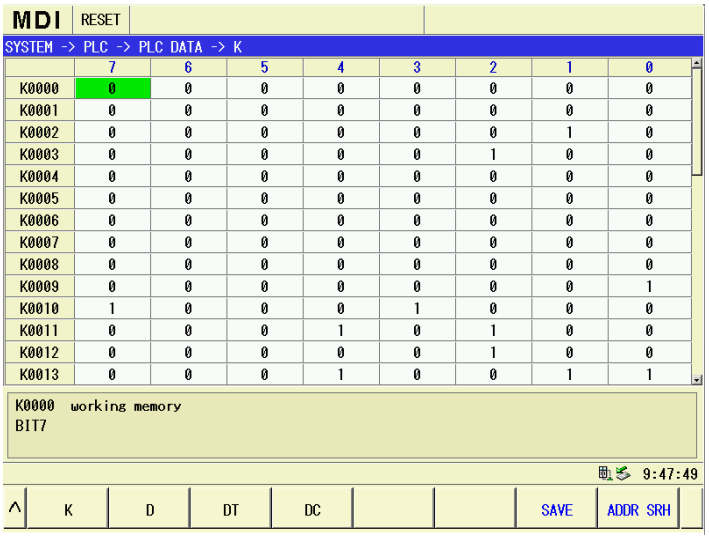


Fig. 1-16

(1) The save of the PLC data

Select the PLC parameter type to be operated by [K], [D], [DT] and [DC]; write the corresponding PLC parameter to its initial value by [SAVE].

Note 1: When the PLC parameter is altered, the modified value is only saved at the system instead of writing to the PLC, therefore, the PLC parameter does not lead out during performing.

Note 2: PLC parameter is lead out together with the PLC when it is performed after pressing the [SAVE].

(2) K parameter setting

In the [PLC DATA] page, enter the K parameter setting display page by [K]. Refer to the above-mentioned figure:

Parameter setting methods:

① Press the [◄], [►], [▲], [▼], [◄] and [►] to select the desired parameter state bit to be modified; or press the [ADDR SRH] to input the desired K variable to be selected, and the press the [OK] softkey, the cursor is then positioned at this parameter. The meaning indicated from this state bit is displayed at the bottom of the screen.

② In the K variable state bit, repeatedly press the [INPUT] to shift between 0 and 1, and then

modify the status of the K parameter state.

③ Move the cursor to complete the modification by pressing the ⇧, ⇩, ⇦ or ⇨.

(3) D parameter setting

In the PLC DATA page, enter the D parameter setting display page by the D. Refer to the following figure.

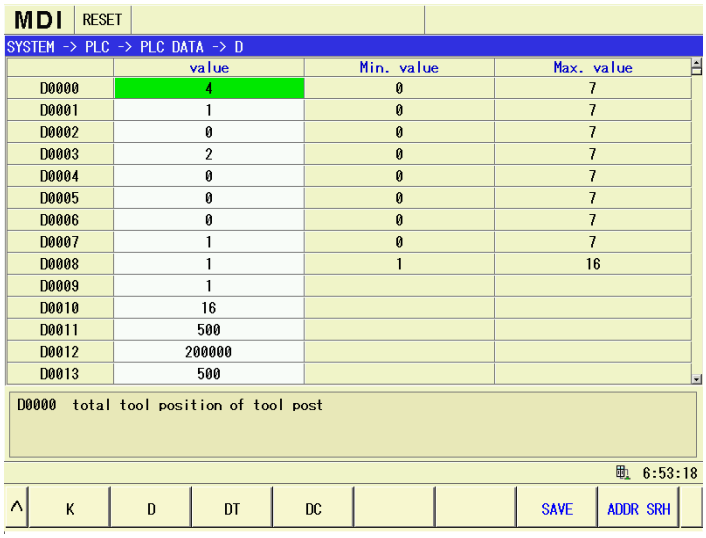


Fig. 1-17

Parameter setting methods:

① Select the D parameter to be modified by ⇧ and ⇩; or press the ADDR SRH softkey, and then input the desired D parameter, press the OK softkey; finally, the cursor positions at this parameter. The meaning described from this parameter displays at the bottom of the screen;

② Press the INPUT, so that the selected D parameter is at the modifiable state.

③ Input the modified numerical value, and then complete the alteration after pressing the INPUT.

(4) DT parameter setting

In the page of PLC DATA, enter the DT parameter setting display page pressing the DT.

The parameter setting method: It is identical with the setting of the D parameter.

(5) DC parameter setting

In the page of PLC DATA, enter the DC parameter setting display page by DC.

The parameter setting method: It is identical with the setting of the D parameter.

1.3.4 PLC On-line Editing

In the ladder diagram page set, press the [>], then the [PROGRAM] —> [LOCAL] in turn, and then enter the ladder diagram program list page; select the desired program to be edited by [⇧], [⇩], and then enter the editing page by [OPEN]; in this case, compile the ladder diagram, symbol table, information display table and the initialization data table.

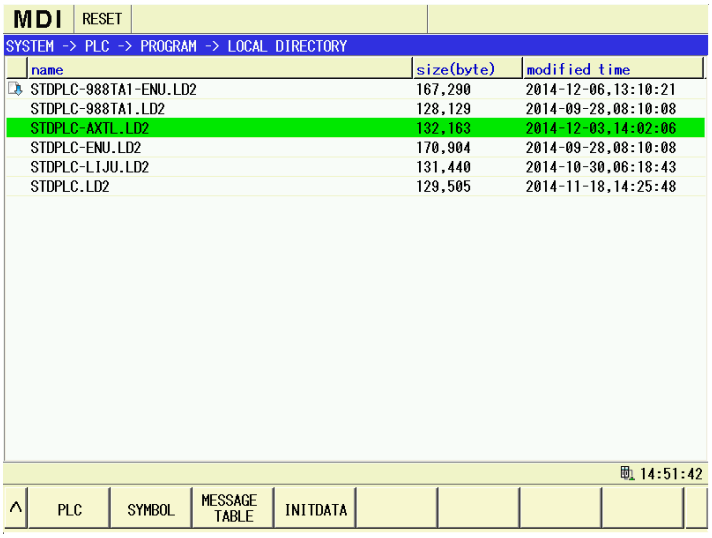


Fig. 1-18

(1) **The checking and editing of ladder diagram**

In the local list page, enter the ladder diagram display and editing window display page by [PLC]; refer to the following figure:

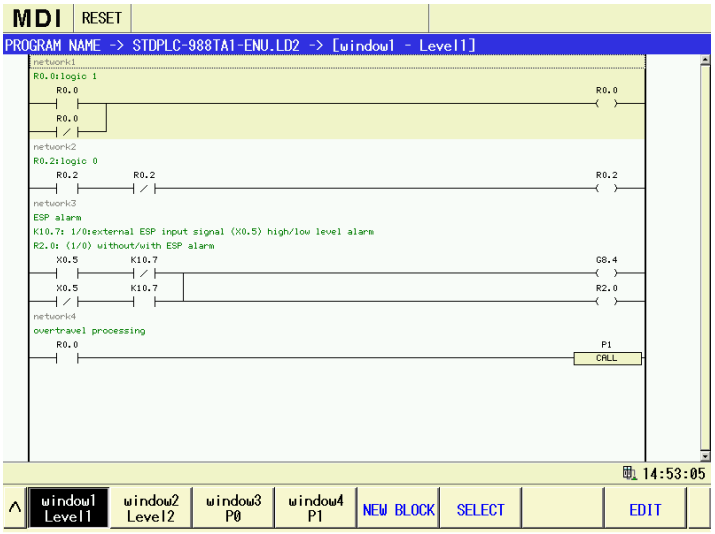


Fig. 1-19

The place where the cursor positions in the figure is indicated by dashed frame, the network area background color of the current cursor is darker than the window background color.

Separately press the corresponding softkeys [window1 Level1], [window2 Level2], [window3 P0] or [window4 P1], and the corresponding blocks are respectively displayed on the window 1, 2, 3 or 4. The block name of current window will then display at the top of the screen.

① To select a block for the window

Firstly, press the corresponding softkeys of the [window1 Level1], [window2 Level2], [window3 P0] or [window4 P1], and then select that of the window should be modified its block; lastly, the corresponding block can be respectively selected for the windows 1, 2, 3 and 4 by [SELECT].

Select the corresponding program block for the window by  ,  ,  or  , confirm the selection and then return by [OK]. In this case, the address of the corresponding block is displayed in the window softkey. For example, [window1 Level1] means that the window 1 is corresponding to the Level 1 block; that is, when pressing the [window1 Level1] softkey, the content of the Level 1 block will then display on this window.

② Create a new block

Press the corresponding softkeys in [window1 Level1], [window2 Level2], [window3 P0] or [window4 P1], select a new window to be created, and then press the [NEW BLOCK] to input the new block name; a vacant block is then created by [NEW BLOCK].

③ Program editing

Select the desired window block program to be edited or modified, enter the editing program page by [EDIT] (Refer to the Fig. 1-20); display the extension softkey by [>] (Refer to the Fig. 1-21).

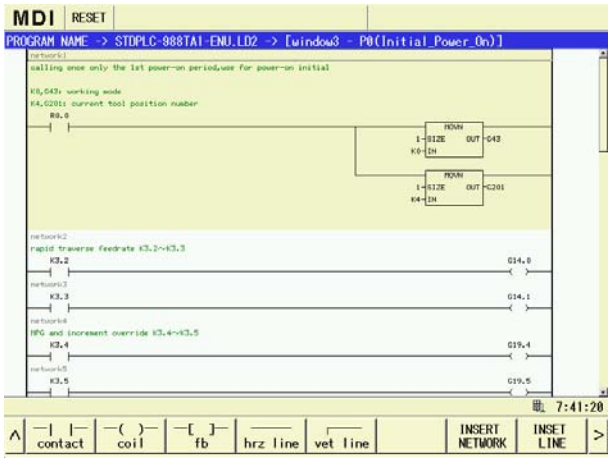


Fig.1-20

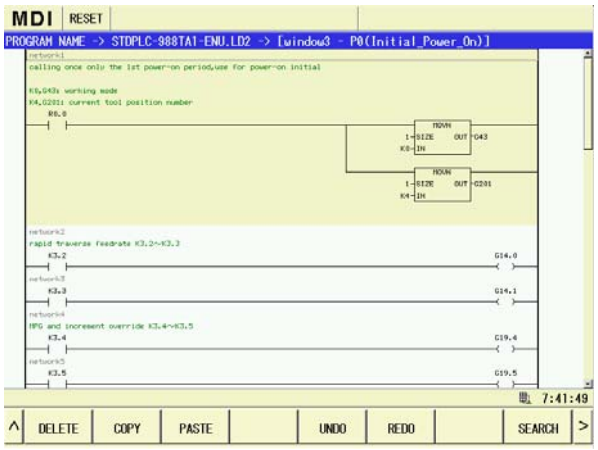


Fig.1-21

A. Move the cursor to the line where to be modified by  ,  ,  or  ; move the cursor to the place where to be operated the component by  or  .

B. Press the [INSERT NETWORK], and then insert a network before the one the cursor locates.

C. Press the [INSET LINE], and then insert a new line after the one the cursor locates.

For example:

Press the [—| |— contact], the screen display is as follows. In this case, the cursor lies in the “Type” selection frame; shifting the Normally Open (N.O.) or Normal Close (N.C.) contactor by  ,  or  ,  .

Press the [CHANGE] to shift the “Address/Symbol” editing frame by cursor, input the address/symbol, and then confirm the completion of the input by [OK] or the “INPUT” button.

Press the [—( )— coil] softkey, the operation is same as the [—| |— contact].

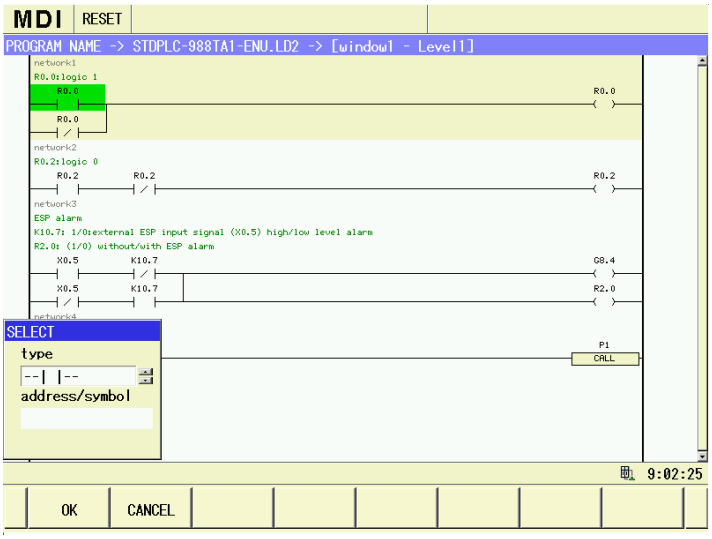


Fig. 1-22

Press the fb softkey, the display screen is as follows:

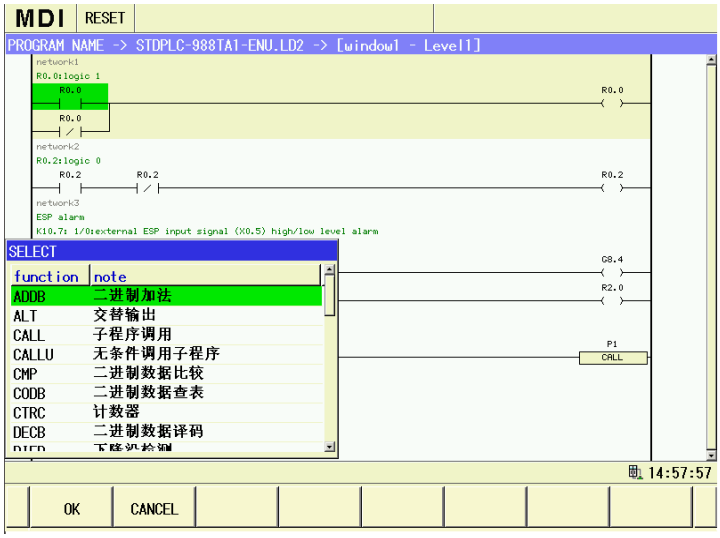


Fig. 1-23

Select the function command to be inserted by ⇧, ⇩, refer to the above-mentioned figure ADDB (Binary Addition); press the OK softkey or the INPUT button, the display page is as follows:

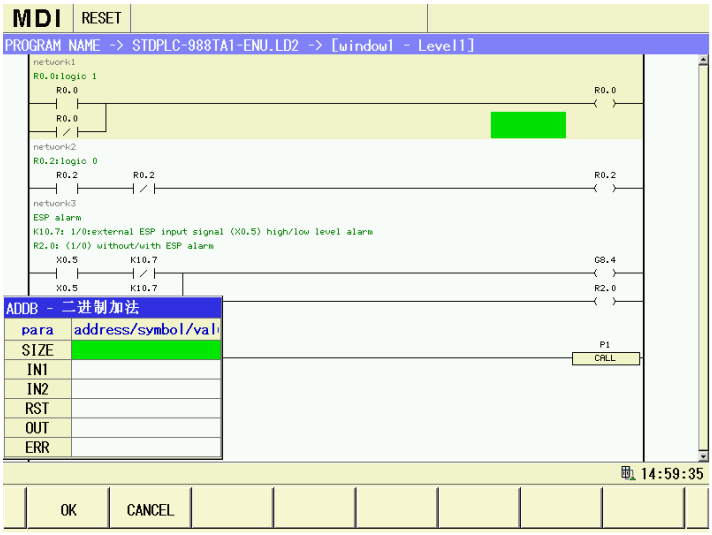


Fig. 1-24

Shift the selection editing frame up and down in each compiling frame by ⇧, ⇩; input the address or data, and then confirm the modification by INPUT. Press the OK to affirm the completion of the editing after the overall editing frames are perfored.

The input editing operations of other function commands are identical with the above-mentioned.

At present, the selected component, line or network can be performed the delection, the copy and the paste operations. As well, you can retreat from the last operation or perform the last retracted operation again.

Shift the searching page by the SEARCH, inspectively press the ADDR SRH, INSTRUT SRH or NETWORK SRH and input the related parameter, command or network, and then position the cursor to its place; also, the cursor can be positioned at the beginning or end of the block.

The function of pressing the DELETE softkey is identical with the one of the SEARCH.

The function of pressing the COPY softkey is identical with the one of the SEARCH. Move cursor to the place where to be operated the network, line and component by ⇧, ⇩, and the paste by PASTE.

Retreat from the prevous operation by UNDO, up to 20-step operations can be memorized

The previous retracted operation can be recovered by REDO.

(2) **The checking and setting of the symbol table**

In the ladder diagram editing page, enter the symbol table display page by SYMBOL:

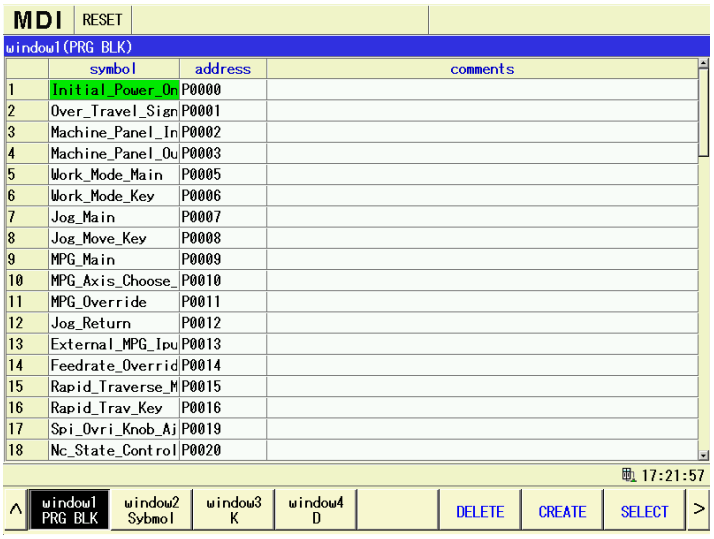


Fig. 1-25

① Respectively press the window1 PRG BLK, window2 Sybmol, window3 K or window4 D to select the symbol information to be displayed on screen. In this case, the name and the one of the corresponding symbol table displayed for the current window are shown at the top of the screen.

Press the SELECT to select the corresponding symbol table for each window.

Create a new symbol table by CREATE and display (Notice: If there is vacancy window, it is displayed at the vacancy one when the new symbol table appears) at the current window. If you want to display the original symbol table in this window, press the SELECT, and then select the original program table.

Display the block symbol table by window1 PRG BLK in the above-mentioned figure, the corresponding address table of the program symbol displays in the block table.

② Display the “Symbol” table by window2 Sybmol; refer to the following figure:

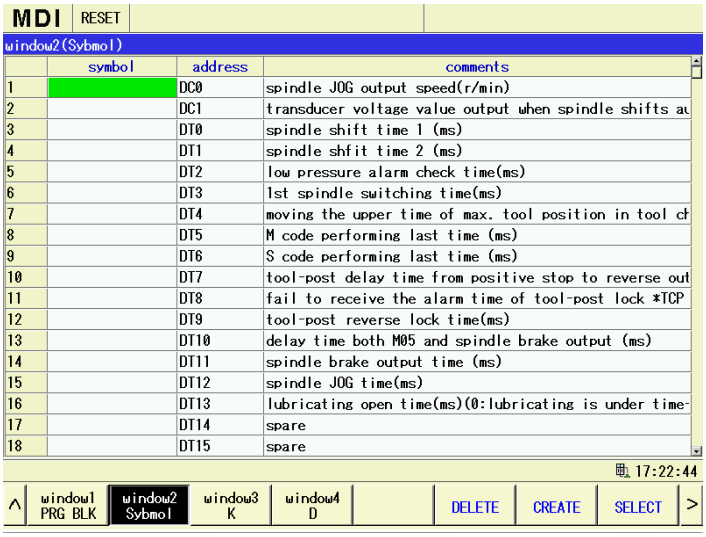


Fig. 1-26

The note meanings of the parameter addresses are displayed X, Y, DC, DT, T and R in the "Symbol" table.

Select and view each parameter address by , , , , , or .

Delet the selected symbol table by DELETE.

In this page, display the extension softkey by >.

Input the parameter address to be searched by SEARCH, and then position the cursor to this address.

Press the INSERT LINE, insert a vacancy line under the one of the cursor postioned.

Press the DELETE LINE, delet the current line of the cursor positioned.

③ The interface and operation of the window3 K, window4 D are identical with the window2 Sybmol.

④ Modify or edit the symbol table (Block symbol table can not be modified here)

Select the window symbol table to be modified, and then select the desired symbol, address or note column by , , , , , or ; press the INPUT, the selected column becomes to the importable state, and then input the symbol, address or note, lastly press the INPUT to complete the modification.

(3) Display the checking and modification of the information table

In the editing page, enter the display information table page by MESSAGE TABLE. Refer to the following figure.

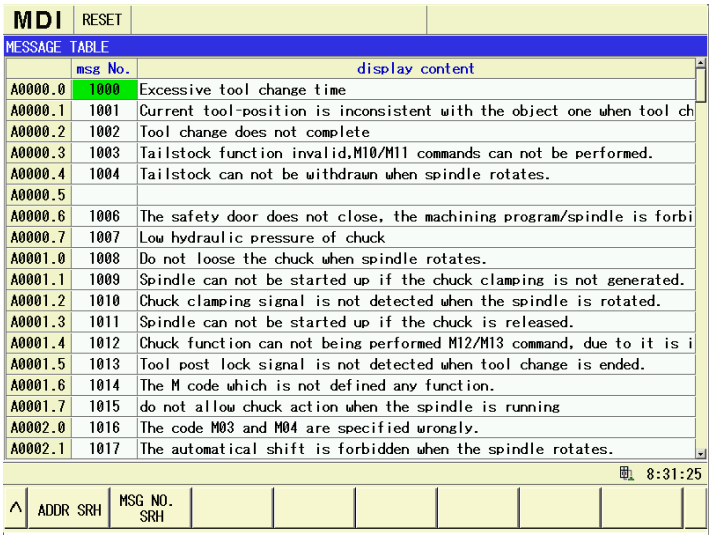


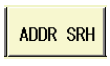
Fig. 1-27

The PLC alarm information A address, the corresponding information number and its corresponding displayed information content are shown in this display information table. Select and check each address, information number and its corresponding display information by  ,  ,  ,  ,  or  .

① The modification of information number and displayed content:

Select the information number or the displayed content to be modified by  ,  ,  ,  ,  or  , which is on the alterable state by  , and then input the information number or the displayed content to be altered, lastly press the  to complete it.

② Address, information number searching:

Input the address or information number to be searched by  or  ; press the “OK” button to search and position the cursor at the address or information number where has been searched.

⑷ The checking and setting of the initialization data table

In the editing page, enter to the initialization data table display page by  .

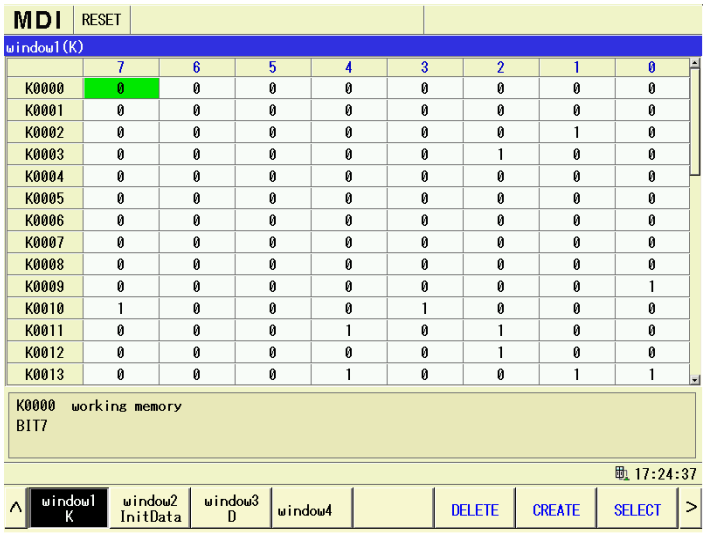
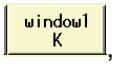


Fig. 1-28

① The setting of K parameter

Select the window 1 by  , the corresponding display is the detailed information of the K parameter, refer to the above-mentioned figure:

Select one position of the corresponding with the K parameter to be modified by  ,  ,  ,  ,  or  , and then the concrete meaning expressed the corresponding position of the K parameter at the current cursor place is displayed at the bottom of this page.

Repeatedly press the  , so that the selected position can be shifted between 0 and 1.

② Initialization data

Press the  , enter to the display page of the corresponding InitData table of the window 2.

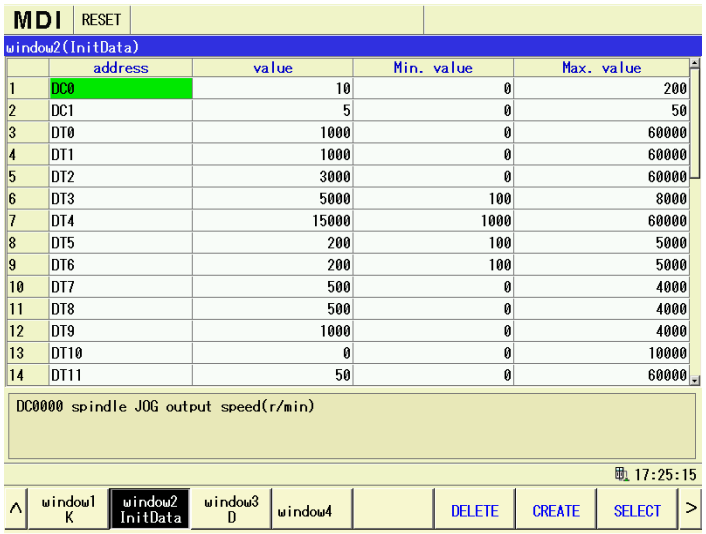


Fig. 1-29

③ The modification and compiling of the data table:

Select the desired page by  and  , select the address value or the numerical, maximum and minimum values to be modified by  ,  ,  or  , and therefore, the selected values are turned into blue background which can be alterable state by  , and modify the value data by number buttons and ESC, and then confirm the modification by  again.

Note: The modification and compilation of the initialization data table are identical with the checking and setting of the symbol table; refer to the (2) The checking and setting of the symbol table in the Section 1.4.3 for details.

1.3.5 PLC Program Transformation

Only the autority, above the machine tool factory level (level 2), can be performed the PLC program transmission.

There are two transmission methods for the PLC program:

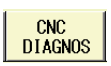
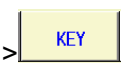
1. Transmission can be performed by GSKComm-M; refer to the Section 1.8 for details (Usage of the PC communication software GSKComm-M).

2. Either the single PLC transmission can be performed with U disk in the ladder diagram page, or the batch of PLC program transmissions can be performed in the file administration page. Refer to the Section 1.2 (The usage of the D disk) for details.

1.4 System Diagnosis

GSK988TA/988TA1/988TB series system diagnosis contains of edit keyboard, hardware interface, bus state and communication data.

1.4.1 Keyboard Editing Diagnosis

Enter to the information interface by  , press the  —>  to the page of the keyboard editing.

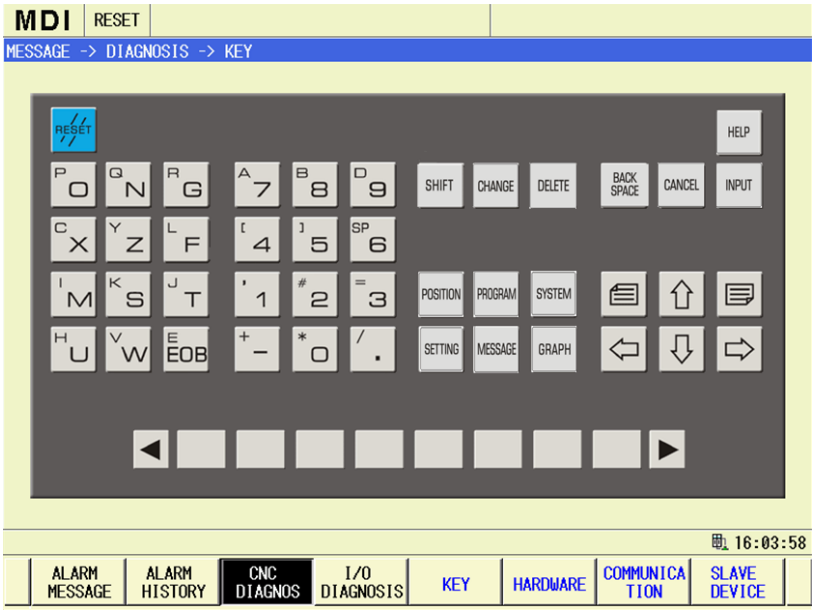


Fig. 1-30

The Edit keyboard diagnosis page displays the system analog editing keyboard, when pressing the button, the corresponding button may display on the other way. In order to prevent the corresponding function operation when checking the diagnosis information for some buttons (such as the direction and page buttons). The current screen can be locked by the LOCK SCREEN.

1.4.2 Hardware Interface Diagnosis

Enter the information interface by MESSAGE, control the CNC DIAGNOS —> HARDWARE entering the hardware interface diagnosis interface.

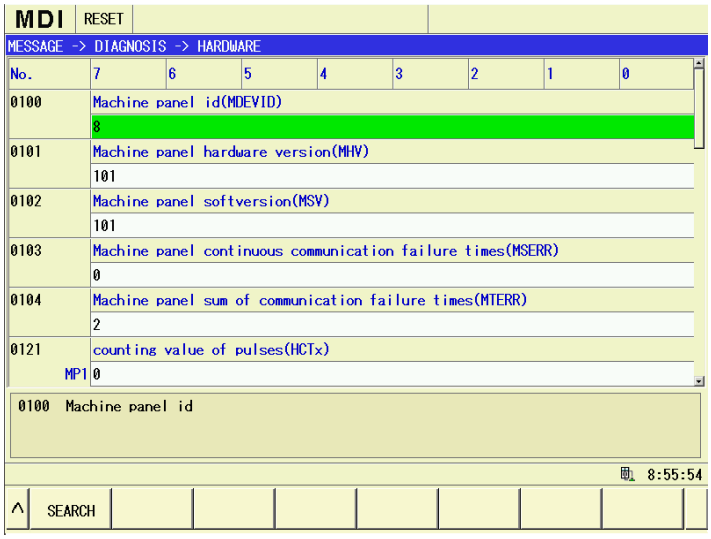


Fig. 1-31

The hardware interface diagnosis page displays the haredware information, hardware error

information and MPG counting for the current CNC.

1.4.3 Bus Communication Diagnosis

Enter the information interface by MESSAGE, and then enter the bus communication diagnosis page by CNC DIAGNOS —>COMMUNICATION.

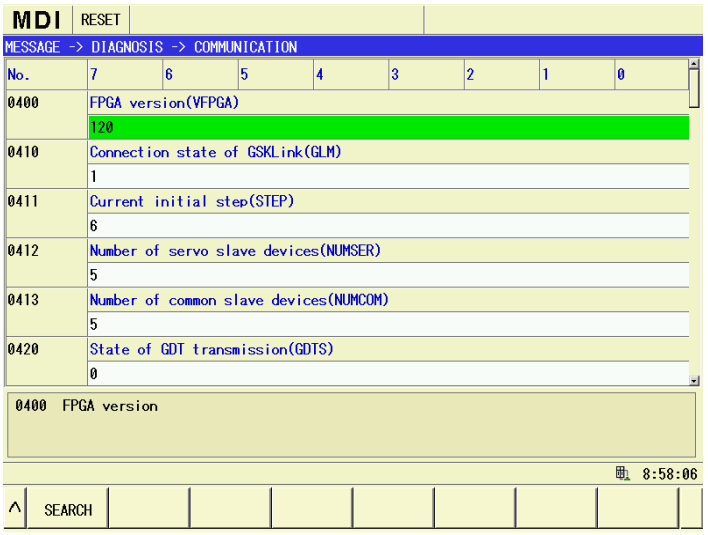


Fig. 1-32

The bus state diagnosis page displays the state information of the current CNC bus connection, which includes the slave numbers of the bus, communication stage and incorrect information of the communication.

1.4.4 Communication Data Diagnosis

Enter the information interface by MESSAGE, and then enter the communication diagnosis page by CNC DIAGNOS → SLAVE DEVICE.

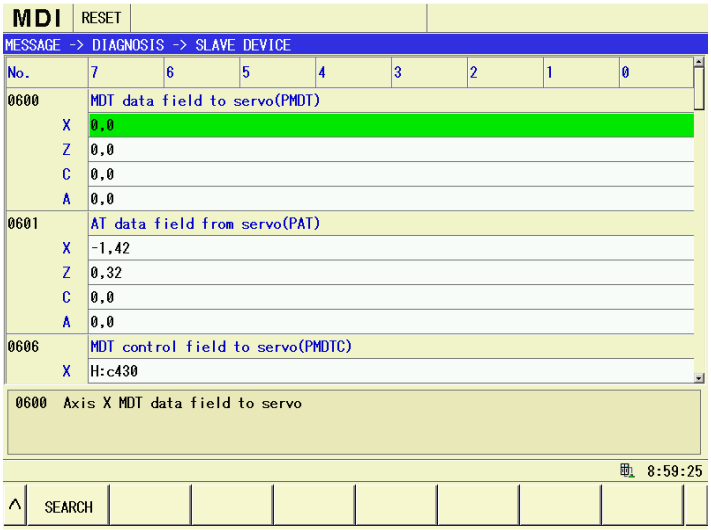


Fig. 1-33

Communication data diagnosis page displays the current real-time data of servo and remote I/O by connecting with bus.

1.5 Servo Adjustment

In the system page set, enter the servo regulation page by GSKLink —> SERVO —> SERVO ADJUST in turn.

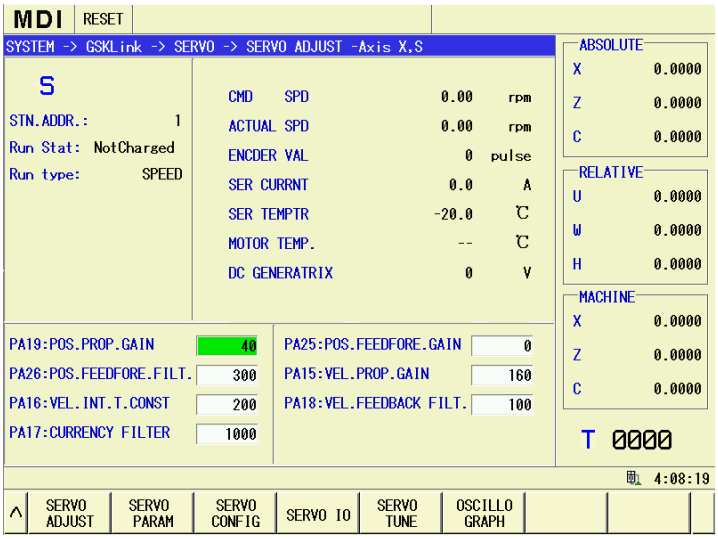


Fig. 1-34

GSK988TA/988TA1/988TB Series servo diagnosis module offers the following functions:

The feedback data by the servo communication is carried out the real-time monitoring for the system control axis, so that the operator can comprehensive the current working state for the equipments, such as the servo, motor (servo logic ID no., operation state, operation mode, command position, feed

The explanations of each data display area on the servo diagnosis interface:

X ：The axis name for current selected axis

STN.ADDR.:：The slave machine corresponding ID number for connecting of this axis

Run Stat:：The current operation state of servo

Run type:：The corresponding servo control method

CMD POS：The servo receives the position pulse number from system (It displays in the position control method)

ACTUAL SPD：The servo receives the command speed value from system (It displays in the position control method)

ACTUAL POS：The position pulse number of servo feedback

ENCDER VAL：Encoder value of current motor's positon

POS ERROR：The offset value of the command position and feedback position

ACTUAL SPD：The actual speed of motor

SER CURRNT：The current working current value of servo

SER TEMPTR：The temperature measure value inside the servo

MOTOR TEMP.：The controlled motor temperature of corresponding servo

DC GENERATRIX：The current DC bus voltage value of servo

Axis shifting: The servo parameter displays shifting among the X axis, Z axis and S axis by pressing the [Axis X], [Axis Z] and [Axis S].

Note: The servo parameter modified at this page does not save, therefore, it will be lost after the servo is turned off. If you want to save it, press the【SAVE】button in the servo parameter page.

1.6 System Debugging

In order to debug the machine tool, GSK988TA/988TA1/988TB Series CNC will classify the frequently-used parameter, which is called the Syste Debugging Function.

Note: The parameter classification in the system debugging function and the different system software version of its conetent may inconsistent, it is subject to the current version.

Enter the setting page set by SETTING function button; In the CNC setting page, press the SYSTEM ADJUST. softkey to enter the system debugging page; refer to the following figure.

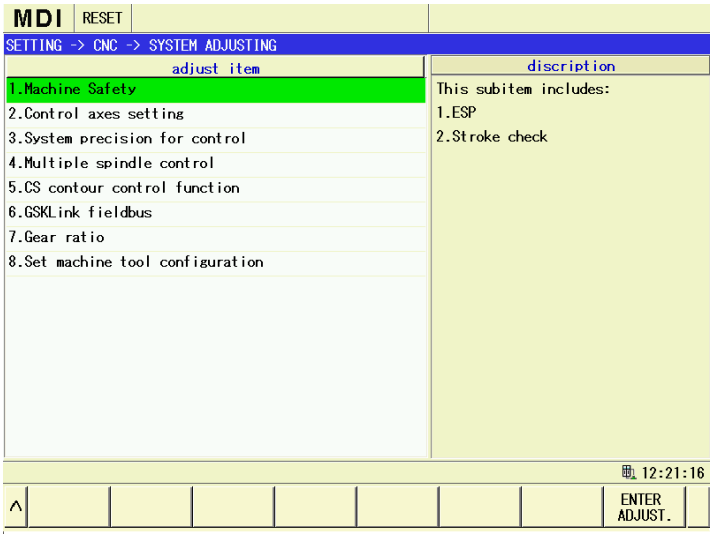


Fig. 1-35

The classification of the system debugging displays at the left of this page, and the item explanation shows at the right side.

The debugging items are divided into: Machine tool safety defense and external switch, Feed axis function setting, Feed axis unit setting, Spindle function setting, CS axis outline control axis function, GSKLink bus setting, Electric gear ratio setting and the relevant setting of the servo control axis, etc. Select the desired debugging items by ⇧ or ⇩, and then enter this debugging item by ENTER ADJUST.

1.6.1 Basis Function Debugging

The basis function debugging contains of the Machine tool safety defense and external switch, Feed axis unit setting, Spindle function setting and GSKLink bus setting. In the debugging page, move the cursor to the “Machine tool safety defense and external switch”, and then enter the debugging page by ENTER ADJUST.

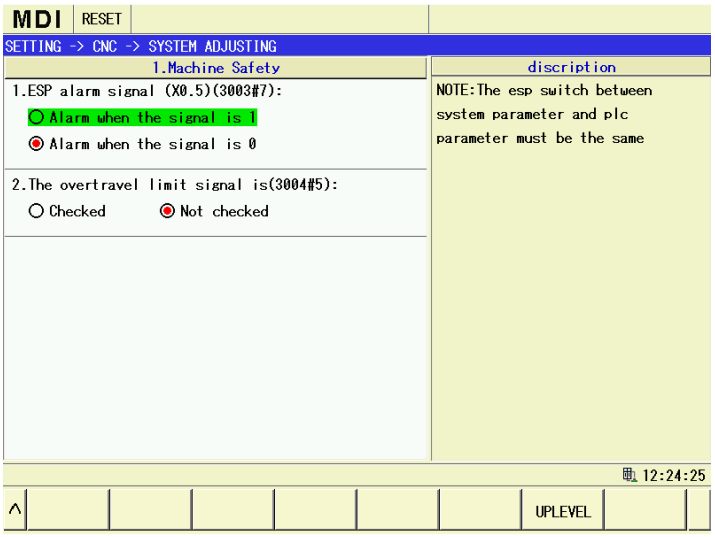


Fig. 1-36

When the parameter switch is turned on and the relevant parameter authority allows as the above-mentioned figure, move the cursor the the corresponding option, and then press the INPUT softkey, the current option is selected accordingly. Return to the last page by UPLEVEL.

1.6.2 AF (Advanced Function) Debugging of System

The AF (Advanced Function) debugging of system includes the CS outline controllable axis and multi-spindle function. The concrete setting method is identical with the above-mentioned basis debugging function.

1.6.3 Electric Gear Ratio Setting

The automatic calculation and setting function of gear ratio are existed at the system debugging function, move the cursor to the “Electric gear ratio setting” in the page of the system debugging item, and then enter to the setting page of the electric gear ratio by ENTER ADJUST.; refer to the following figure:

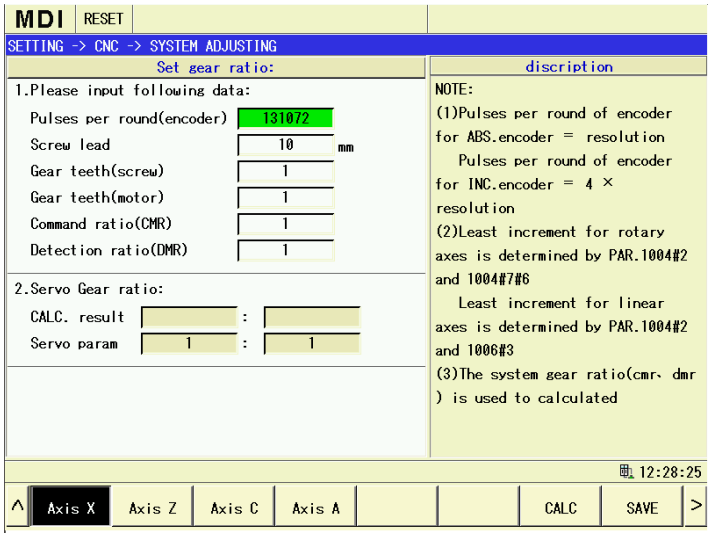


Fig. 1-37

Controllable axis selection: Select the different axes by the various axis selection softkeys.

（1） Data input

Pulses per round(encoder): Set the pulse number for one circle of the encoder

Pulse number of the absolute motor encoder one-turn = Encoder linear of motor

Pulse number of the incremental motor encoder one-turn = 4 x Encoder linear of motor

Note: If the GSKLink bus of the setting axis is normally connected, the system may automatically read the pulse number of encoder instead of setting by hand.

Screw lead: Set the pitch of the ball screw, the pitch of the rotation axis is fixed at 360°.

Gear teeth(screw): If the motor and lead screw is directly connected, set the gear number of the ball screw port as 1.

Gear teeth(motor): If the motor and lead screw is directly connected, set the gear number of the motor port as 1.

（2） Calculation result

Servo Gear ratio: The system will automatically calculate the electric gear ratio of the current axis by CALC.

（3）Gear ratio calculation

Select the desired axis to be set and set the the leading, mechanical driving ratio and one-ture pulse number of encoder, and then control the CALC softkey, the system will be wrought out at the gear ratio based upon the above-mentioned input parameter and displayed at the calculation result. If the system does not connect the GSKLink bus, the value should be record, and then modify and save the servo gear ratio numerator or denominator parameter on the drive by hand. If the

system GSKLink bus is already connected, directly press the SAVE, the system will be set the calculation result to the corresponding servo drive and then save it.

Note: The calculated value from the gear ratio operation function is already considered the CNC gear ratio and the parameter setting of other relevant gear ratio at the current system.

1.6.4 One-Touch Backup & One-Touch Recovery

Above the 2 levels of the operation authority, two functional buttons may occur; refer to the "One-touch recovery" and "One-touch backup".

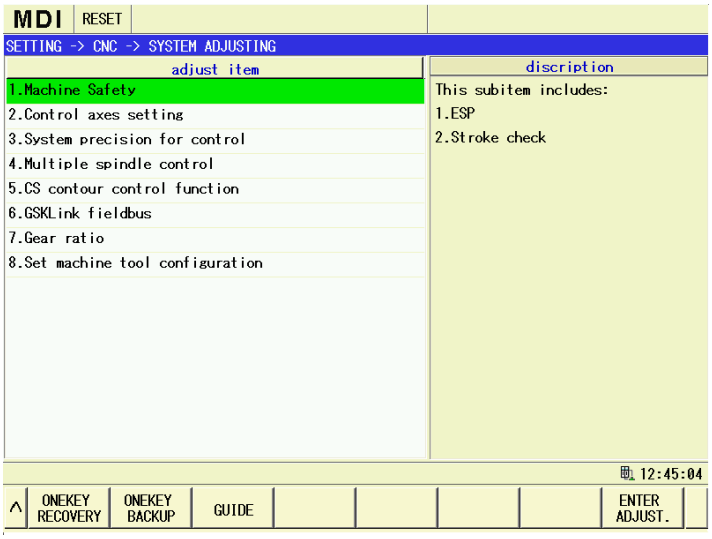
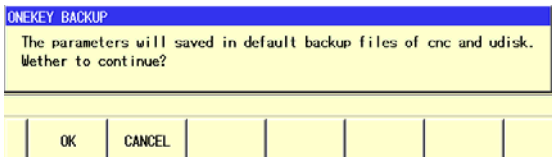


Fig. 1-38

One-touch backup: One-touch backup is copied the system parameter, servo drive parameter, PLC program and PLC parameter to CNC memory and U disk.

Insert the U disk to the CNC, press the ONEKEY BACKUP button in the system debugging page, the system will show the 

dialog frame; you can backup a copy of file used by CNC separately in system and U disk by OK; the other operations can be performed by QUIT DIALOG after the backup executes; refer to the following figure.

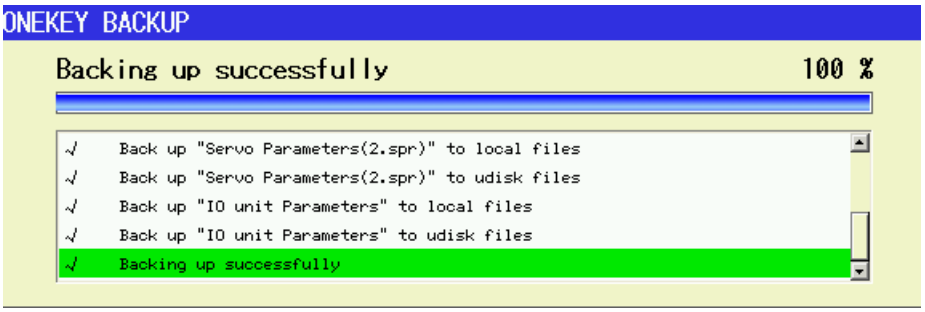


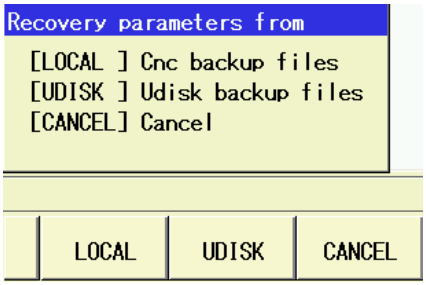
Fig. 1-39

Note 1: A fold named by "GSK988TA_CONF" can be generated in the root directory of the U disk when U disk is performed the backup.

Note 2: If the U disk is already performed the backup before for the other systems, if you want to use this U disk again, the following file will cover the previous one, and therefore, different machine tools should be used different U disk when performing a backup.

One-Touch recovery: One-touch recovery is recovered the previous backup files such as the system parameter, servo drive parameter, PLC program and PLC parameter at the current CNC.

That the U disk is performed "one-touch backup" inserts to the CNC, in the case of the parameter switch is turned on, press the ONEKEY RECOVERY in the system debugging page, and then the resource of parameter is recovered after pressing OK in the dialog frame sprang out from system; refer to the following figure.



Local: Recovery the previous backup CNC parameter.

U disk: Recover the file in the U disk to the current CNC, and then use it.

Cancle: Cancel the current operation

The recovery is selected from U disk after pressing the UDISK. The following figure is shown the successful recovery, in this case, parameters are already recovered and can be normally used after the power is turned on again.

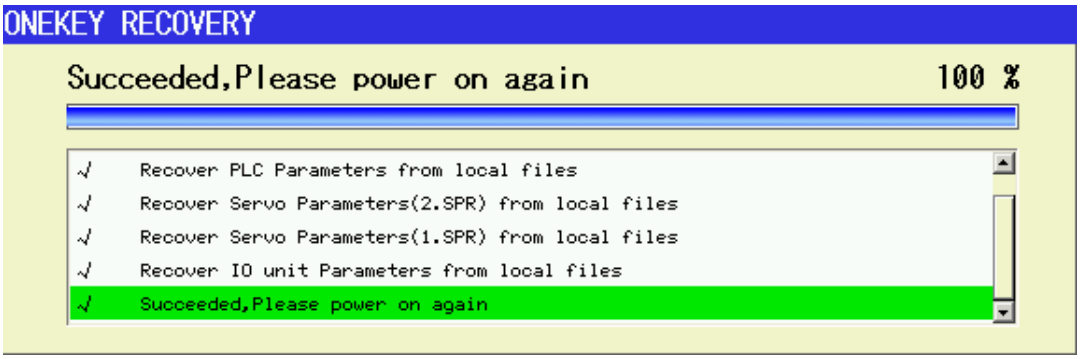


Fig. 1-40

Note 1: It is necessary to guarantee the folder named with "GSK988TA_CONF" at the root directory of U disk, and the file in the folder is kept a backup from the previous one.

Note 2: It is suggested to use one-touch recovery function in the same condition of the machine tool configuration, if it uses at the different configuration machine tools, the recovery may fault due to some parameters are unmatched.

1.6.5 Debugging Guiding

The different parameter conditions should be set based upon the different machine tool configurations after the power is turned on, in order to convenient to debug, press the SETTING function button to enter the page set; in the CNC setting page of this page set, press the SYSTEM ADJUST. softkey to enter the system debugging page, and then enter the debugging guiding page by pressing the GUIDE;

After the setting is performed step by step, it is convinent to complete the setting accordingly.

1.7 Servo Debugging

After the GSKLink bus communication connection of the servo is successfully connected, observe the motor's speed fluctuation or roundness testing figure by servo debugging function at the side fo the system, judge whether the current servo parameter is reasonable and optimal.

Enter the servo debugging function by GSKLink → SERVO → SERVO TUNE in the system page.

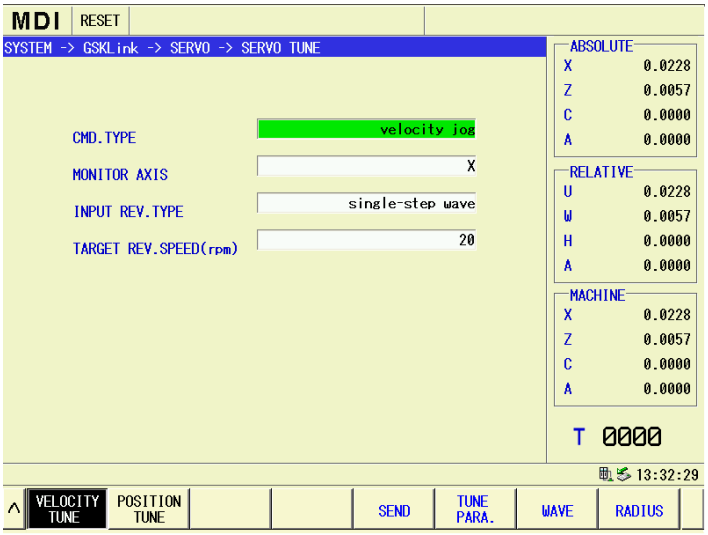


Fig. 1-41

1.7.1 Velocity-Loop Testing

In the MDI method resetting state, deliver the velocity speed control command to the drive by SEND, the motor may always rotate keep pressing; the delivery may stop until releasing; press the WAVE softkey, judge whether the servo parameter is set reasonably by observing the fluctuation figure.

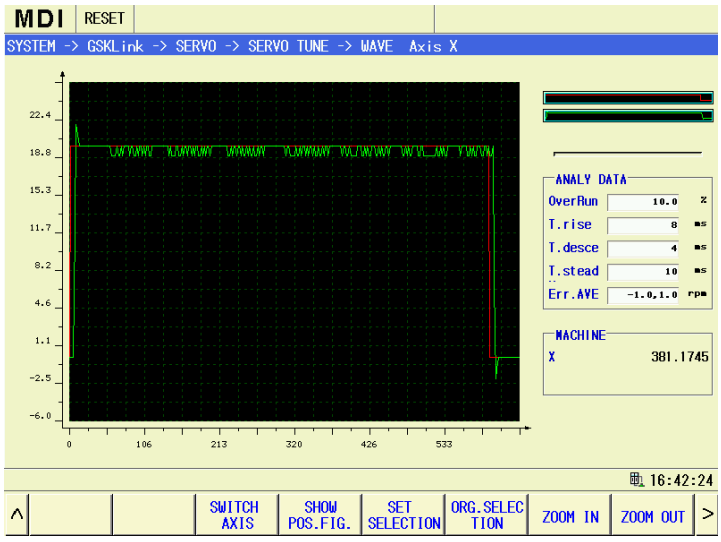


Fig. 1-42

1.7.2 Position-Loop Testing

1.7.2.1 Position JOG

In the position loop testing interface, press the SAMPLE and then perform the movement

command by cycle start button; press the WAVE softkey after the system is executed, check the speed or position fluctuation figure, judge whether the parameter is set reasonably based upon the fluctuation figure, if does not, the parameter should be altered accordingly.

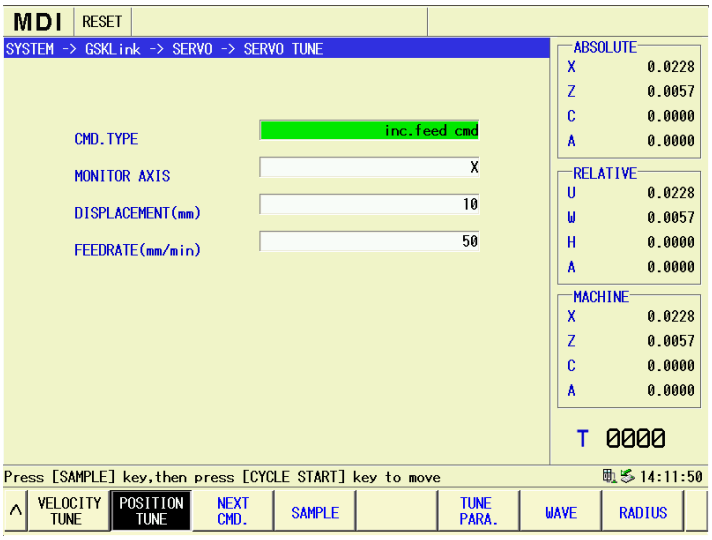


Fig. 1-43

1.7.2.2 Roundness Testing

In the position loop testing page, press the NEXT CMD., the shifting can be performed between the position JOG and the roundness testing, after shifting to the roundness page; press the SAMPLE and then execute the movement command by pressing the cycle start button; press the WAVE to check fluctuation figure, alternatively, press the RADIUS to view the roundness figure after the system is performed; adequately adjust the parameter based upon the fluctuation in the figure.

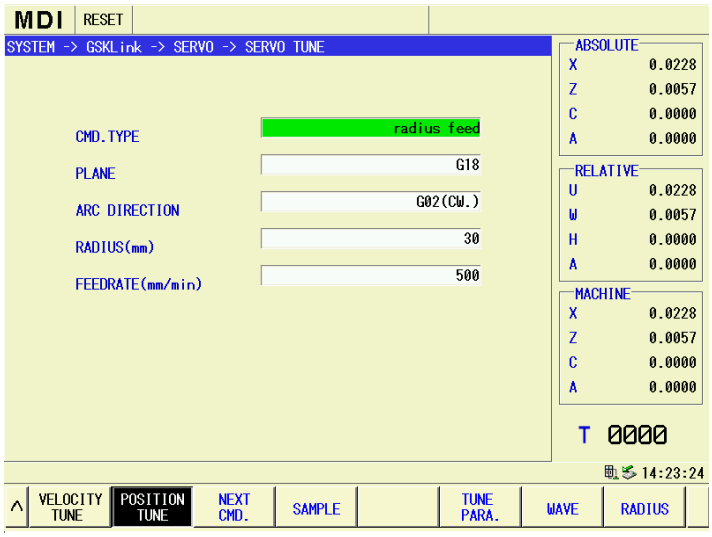


Fig. 1-44

1.8 Use of PC Communication Software GSKComm-M

This section that is simply described the GSKComm-M function may be used only in the machine tool debugging by the manufacturer. Refer to the *GSKComm-M User Manual* in the software CD for details.

GSKComm-M is the communication administration software especially for the machine tool manufacturer; refero to the following interface. It can be carried out the functions such as the loading/downloading of the file between the PC and CNC, DNC communication, the editing of the CNC parameter, the administration and editing of the component program, the observation of cutter & screw compensations and the ladder diagram compiling. It also owns the simple operation, high communication efficiency and reliability.

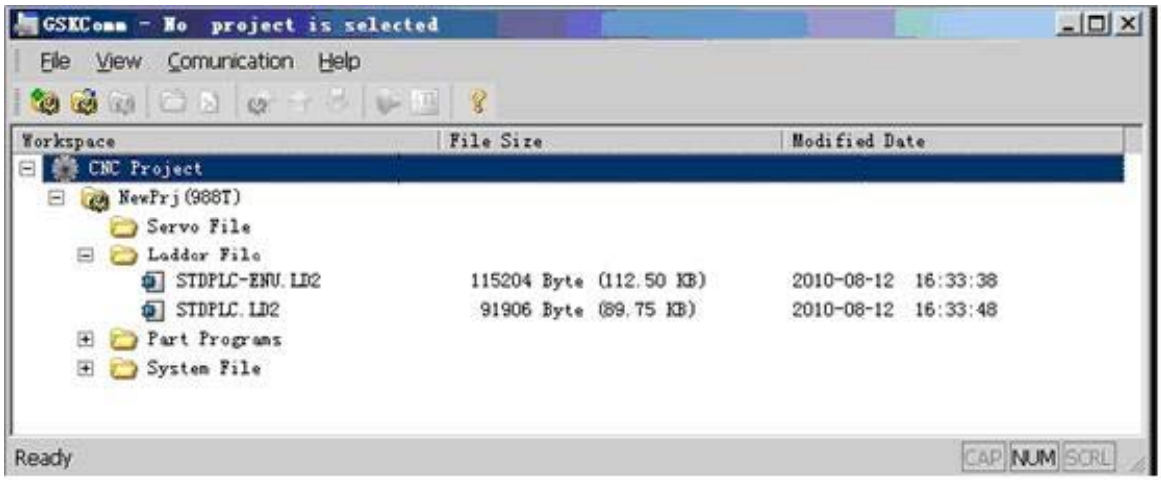


Fig. 1-45

1.8.1 Preparation Beform GSKComm-M Communication

（1） Network connection

The connection between PC and CNC:

One port connects the 988TA access; the other one connects the Computer or Router whith the common reticle.

IP setting of CNC:

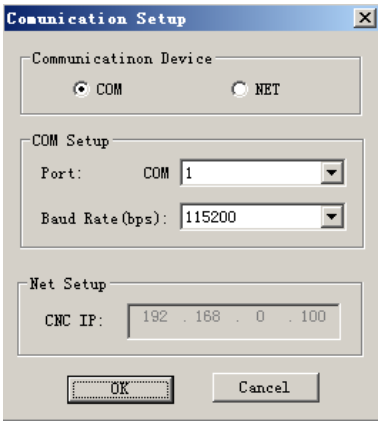
Press SETTING —> CNC SETTING —> ETHENET in turn, set the IP address and gateway, etc. after entering the system IP setting page.

IP setting of GSKComm-M for PC:

Clik the menu by the left mouse key after operating the communication software, select the “Communication－> Communication setting”, the interface display as the right figure:

Communication setting: select the “Network”.

Network setting: Write the IP address in CNC.



（2）Authority setting

User should set the corresponding authority when loading or downloading by GSKComm-M; otherwise, the operation may fail.

PC download data	The lowest limit of the CNC	Remark
PLC file	Level 2	
Parameter	Level 3	Open the parameter switch
Component program	Level 3	Open the program switch
Macro variable	Level 4	Open the program switch
Tool offset value	Level 4	
Screw compensation data	Level 5	Open the parameter switch
Tool life-span file	Level 5	

Table 1-2

1.8.2 File Downloading（PC－>CNC）

GSKComm can be delivered the overall files CNC once, as well the single one.

(1) File addition

Firstly, click and select the file type to be added (for example, system file, component program and ladder diagram figure file).

Secondly, click once the  or the right key, select the “File addition”, and then its dialogue frame (refer to the following left figure) springs out; select the file to be added (Several items can be selected by “Shift”), finally, the addition is executed by “Open”.

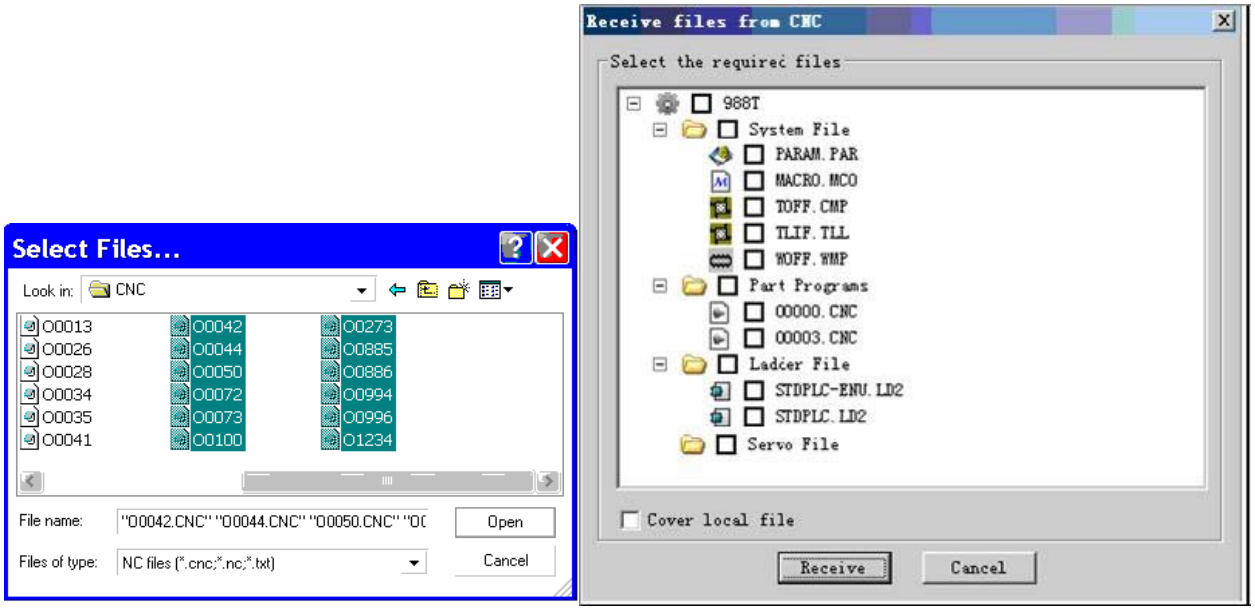


Fig. 1-46

(2) Multiple files download

First, select the desired project to be transported;

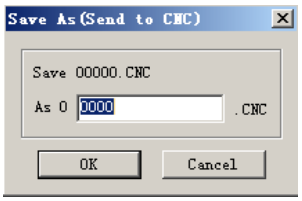
Second, click the  or the right key, select the “deliver the project to CNC”, that is, the “Deliver the file to CNC” dialog frame appears (Refer to the above-mentioned left figure).

In this dialog frame, click the left option of the file name to select the desired file to be delivered. File name “—>” means that the one is saved inside the CNC, and the file name can be modified and saved by double-click.

Lastly, click the “Delivery”, the selected file can be sent to the CNC with the corresponding saved name.

(3) Single file download

Select the desired file to be downloaded, click the  or the right key, then select the “Deliver the file to CNC”, and then the dialog frame appears: The file name saved in CNC can be altered in this dialog frame.



Click “OK”, and then transmit the file to CNC.

1.8.3 File Uploading（CNC—>PC）

First, select a project.

Second, click the  or select the menu “Communication —> File acceptance from CNC”, spring out the dialog frame of “File acceptance from CNC” (refer to the following left figure); Select the desired file to be uploaded, and then click the “Acceptance beginning” button; the “Browsing file

folder" dialog frame is then displayed (refer to the following right figure):

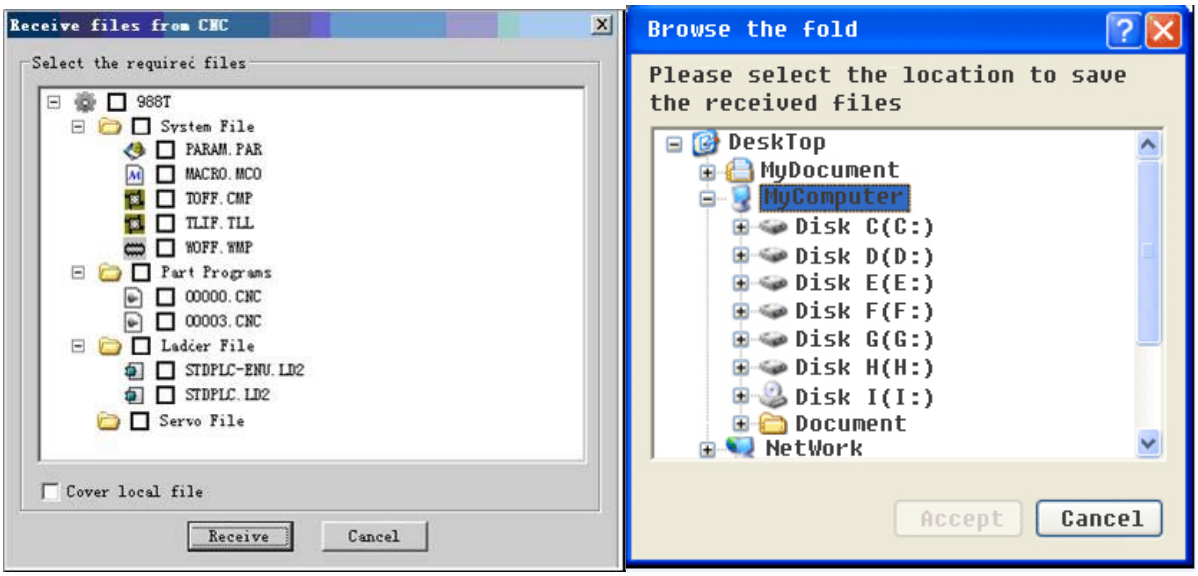


Fig. 1-47

Select the folder to be saved of the uploading file, click the "OK", the file uploading begins from CNC.

Chapter 2 Machine Tool Debugging & Function

2.1 GSKLink Bus Connection

GSK988TA/988TA1/988TB series and the feed drive unit, spindle driver and extention I/O unit are connected with the GSKLink bus; refer to the Section 1.2.2 “Volumn One Installation Connection” for the concrete figure; it is necessary to set the relevant parameter after connecting each component, therefore the equipment can be normally used, refer to the following table.

Relevant Parameter					
	Type	**Para. No.**	**Bit**	**Parameter Meaning**	**Remark**
CNC	**GSK988TA/988TA1/988TB series parameter**	9000	#0	Wheather the system GSKLink communication is enabled 0：Disabled 1：Enabled	
		1023		Logic ID number along each axis	The set axis number corresponds with the setting value of the drive unit along each axis.
		3050		The I/O quantity controlled by system	
		3051		Logic ID number of the system control I/O unit 1	
		3052		Logic ID number of the system control I/O unit 2	
		3053		Logic ID number of the system control I/O unit 3	
		3054		Logic ID number of the system control I/O unit 4	
		3717		Amplifier number of each spindle	The spindle non-bus connection is common analog spindle when it is set to -1～-8. It is the bus spindle when setting as 1～99; the setting value should be consistent with the corresponding spindle drive unit.
Servo	**GS2000 series**	PA156		GSKLink communication servo slave number	It is corresponding to the logic ID number of each axis with the CNC parameter 1023.
		PA4		Control method selection: 21 is the bus method	
	300 0 seri	PA156		GSKLink communication servo slave number	It is corresponding to the amplifier number of each

					spindle for each axis with the CNC parameter 3717.
		PA4		Control method selection: The bus method is 21.	
I/O Unit	**IOL**				The communication parameter is set by system side, refer to the Section 1.1.3 of “Volumn Two Debugging” for details.

Table 2-1

Note 1: The slave number corresponding to the GSKLink communication can not be repeated; otherwise, the servo and the GSKLink communication of the system can not be established.

Note 2: The slave number in the servo should be modified (It can be enabled after the power is turned on) on the driver by hand before connecting with the GSKLink.

2.2 ESP & Hard Limit

GSK988TA/988TA1/988TB series system owns the software limit function, for the safety’s sake; simultaneously, it is suggested to use the hard limit installing the stroke limit switch with the negative or positive along each axis; the connection figure is shown below:

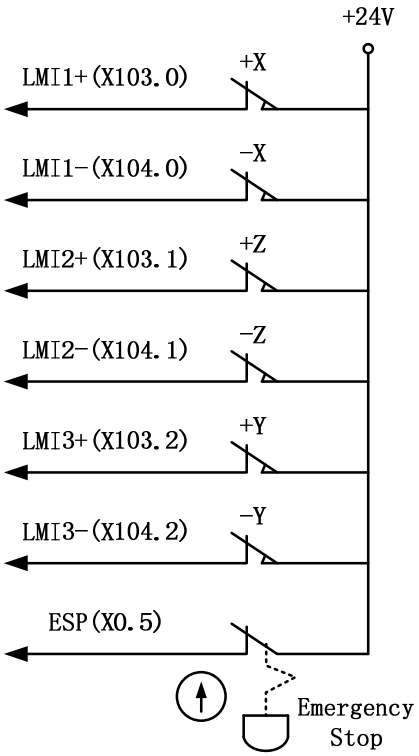


Fig. 2-1

In the MANUAL or MPG mode, slowly move each axis to verify the effectiveness of overtravel limit switch, the correction of the displayed alarm and the validity of the overtravel releasing button, (The axis that moves towards to the negative direction can be released the alarm).

The "ESP" alarm may occur in CNC when pressing the ESP button.

Relevant Parameter				
	Para. No.	**Bit**	**Parameter Meaning**	**Remark**
CNC parameter	**3003**	**#7**	ESP external ESP alarm signal (X0.5) 0:ESP alarm occurs when the input signal is 0 (LOW LEVEL) 1: ESP alarm occurs when the input signal is 1 (HIGH LEVEL)	The two parameters should be regarded as accordance
Standard PLC parameter	**K0010**	**#7**	External ESP alarm signal (X0.5) 0:ESP alarm occurs when it is the LOW LEVEL 1:ESP alarm occurs when it is the HIGH LEVEL	
CNC parameter	**3004**	**#5**	Overtravel limit signal 0: Inspection 1: Not inspection	
Standard PLC parameter	**K0010**	**#2**	The level selection of overtravel signal input 0: Alarms in HIGH LEVEL 1: Alarms in LOW LEVEL	

Table 2-2

2.3 Basis Parameter Setting of Axis

（1） Axis name

The basis axis number of the GSK988TA/988TA1/988TB series turning machine CNC system is 2 axes; the extension controllable axis number is 6 axes (including the Cs axis); the basis linkage controllable axis number is 3 axes.

Relevant Parameter		
Para. No.	**Parameter Meaning**	**Remark**
8130	The overall controllable axis number of the system	
1010	CNC control axis number	This parameter value should be less than the parameter No. 8130.
1020	Programming name of each axis	The axis names of each axis can not be same.
1022	The attributions of each axis in the basis coordinate system	
1023	Servo axis number of each axis	The set axis number corresponds with the driver setting value of each axis.

Table 2-3

（2） The unit of axis

User set the equivalent of the system, input/output uint of each axis based upon their requirements after the setting is performed the above-mentioned parameters, also, set the least input, output incremental value of the axis based on the machining accuracy.

Relevant Parameter		
Para. No.	**Bit**	**Parameter Meaning**
0000	**#2**	Input unit 0: Metric system 1: Inch system
1001	**#0**	The least movement unit of the linear axis is: 0: Metric system (Metric machine tool) 1: Inch system (Inch machine tool)
1004	**#1**	Set the least input unit and command increment 0：0.001mm 1：0.0001mm
1006	**#3**	Set the movement amount along each axis: 0: Radius specification

		1: Diameter specifiecation
3401	**#0**	The address of decimal point can be used, the setting is shown below when the decimal point is omitted: 0: It regards as the least setting unit 1: It regards as mm, inch, sec unit

Table 2-4

（3） The type of axis

In the machining, the used axis is set to linear axis or rotation axis based upon the machining requirements.

Relevant Parameter		
Para. No.	**Bit**	**Parameter Meaning**
1004	**#6, #7**	Set the times between the least command increment of the rotation axis and ISC parameter 00：×1 times .01：×10 times 10：×100 times
1006	**#1, #0**	00: Linear axis 01: Rotation axis (Type A) 11: Rotation axis (Type B)
1008	**#0**	Set whether the cycle display function of rotation axis is enabled. 0: Disabled 1: Enabled
1008	**#1**	Set the rotation direction of the axis by the absolute command 0: The closer detstination of the rotation direction 1: Specify the direction by command value symbol
1008	**#2**	The relative coordinate is: 0: The cycle is performed regardless of the movement amount for each rotation. 1: The cycle is performed based upon the movement amount for each rotation.
1260		The movement amount per revolving of each axis when the rotation axis is performed.

Table 2-5

2.4 Gear Ratio Calculation & Setting

The immediate meaning of electric gear ratio is that the movement distance of the machine

carriage is consistent with the program command value (the distance of the machine tool coordinate). If the axis is radius programming (Radius/Diameter programming setting: Bit 3 of parameter 1006). The actual axis movement distance equals to the machine coordinate movement distance displayed on the system; if the axis is diameter programming, the twice of the actual axis movement distance on the machine tool should be equaled to the machine tool coordinate movement distance showed on the system.

Gear ratio setting includes the gear ratio of CNC and the gear ratio setting of the servo drive unit, it is better to note it during the catual application.

Relevant Parameter		
Para. No.	**Bit**	**Parameter Meaning**
0000	**#2**	Input unit 0: Metric system 1: Inch system
1004	**#1**	The least input unit and least command increment 0：ISB system 1：ISC system
1006	**#1, #0**	Set the linear axis or rotation axis 00: linear axis 01: rotation axis (type A) 11: rotation axis (type B)
1006	**#3**	Set the movement amount of each axis is: 0: Radius specification 1: Diameter specification
1816		Detection multiplication ratio of each axis
1820		Command multiplication ratio of each axis

Table 2-6

Incremental System						
	Input	**Diameter/ Radius programming**	**IS－B**		**IS－C**	
			Least setting unit	**Least movement unit**	**Least setting unit**	**Least movement unit**
Metric machine tool	**Metric system**	**Diameter**	0.001mm	0.0005mm	0.0001mm	0.00005mm
		Radius	0.001mm	0.001mm	0.0001mm	0.0001mm
	Inch system	**Diameter**	0.0001 inch	0.0005mm	0.00001 inch	0.00005mm
		Radius	0.0001 inch	0.001mm	0.00001 inch	0.0001mm
machine	**Metric system**	**Diameter**	0.001mm	0.00005 inch	0.0001mm	0.000005 inch
		Radius	0.001mm	0.0001 inch	0.0001mm	0.00001 inch

	Inch system	**Diameter**	0.0001 inch	0.00005 inch	0.00001 inch	0.000005 inch
		Radius	0.0001 inch	0.0001 inch	0.00001 inch	0.00001 inch
Rotation axis			0.001deg	0.001deg	0.0001deg	0.0001deg

Table 2-7

2.4.1 Gear Ratio Calculation

Gear ratio calculation formula:

$$\text{Gear ratio} = \text{Least command increment} \times \frac{\text{encoder pulses per revoltion}}{\text{Lead}} \times \frac{Z_M}{Z_D}$$

The least movement unit: The least command unit conveyed to machine tool from CNC; also, it is the least increment of the tool movement on machine tool, which is called the least movement unit.

The pulse number of the pulse encoder one-turn = Encoder linear number (the absolute encoder used by feed motor)

= 4 x endoder linear number (the incermental encoder used by feed motor)

ZM: Gear number at the lead screw terminal

ZD: Gear number at the motor terminal

For example:

Machine tool configures as GSK988TA and GS2050C, which uses the ISC system programming; wherein, the X axis is diameter programming, its leading is 6mm; Z is radius programming, its leading is 8mm; the motor is directly connected with the X, Z axes lead screw (ZM: ZD=1: 1) and used the 17-bit absolute encoder (Encoder linear number: 217 (that is 131072)), and then calculate the corresponding gear ratio of the X and Z axes.

The calculation of the X axis：

The least movement unit: 0.00005mm (ISC system, diameter programming)

$$\text{Gear ratio} = \text{Least command increment} \times \frac{\text{encoder pulses per revoltion}}{\text{Lead}} \times \frac{Z_M}{Z_D}$$

$$= 0.00005 \times \frac{131072}{6} \times \frac{1}{1} = \frac{2048}{1875}$$

The calculation of the Z axis：

The least movement unit: 0.0001mm (ISC system, radius programming)

$$\text{Gear ratio} = \text{Least command increment} \times \frac{\text{encoder pulses per revoltion}}{\text{Lead}} \times \frac{Z_M}{Z_D}$$

$$= 0.0001 \times \frac{131072}{8} \times \frac{1}{1} = \frac{1024}{625}$$

2.4.2 Gear Ratio Setting

$$\text{CNC}:\text{Gear ratio}=\frac{\text{Command multiplying ratio (CMR : No.1820)}}{\text{Detect multiplying ratio (DMR : No.1816)}}$$

$$\text{Servo}:\text{Gear ratio}=\frac{\text{Position pulse command multiplying ratio (PA12)}}{\text{Position pulse command frequency division ratio (PA13)}}$$

The top velocity allowed by CNC will be descended when the numerator is more than the denominator of the electric gear ratio (CMR/DMR) at the CNC side. When its numerator is less than the denominator, the position accuracy of the CNC may fall. In order to guarantee the position precision and velocity index of the CNC, it is suggested to set the electric gear ratio of the CNC as 1:1 when the digit servo of the electric gear ratio is matched, and then set the calculated electric gear ratio to digit servo.

For example: (Gear ratio is the value in its calculation examples)

X axis

The setting of the CNC gear ratio

The electric gear ratio of the CNC sets to 1:1, that is, the ratio value of the CMR/DMR is set to 1.

The setting value of the CMR (Parameter No.1820) is 2.

The setting value of the DMR (Parameter No.1816) is 2.

The setting of the servo gear ratio

The electric gear ratio of servo sets to 2048 / 1875.

The setting value of the PA29 is 2048.

The setting value of the PA30 is 1875.

Z axis

The setting of the CNC gear ratio

The electric gear ratio of the CNC sets to 1:1, that is, the ratio value of the CMR/DMR sets to 1.

The setting value (Parameter No.1820) of the CMR is 2.

The setting value (Parameter No.1816) of the DMR is 2.

The setting of the servo gear ratio

The electric gear ratio of the servo is set to 1024 / 625.

The setting value of the PA29 sets to 1024.

The setting value of the PA30 sets to 625.

2.5 Setting and Adjustment of Relevant Servo

2.5.1 Servo Parameter Setting of CNC

After the system and servo are correctly connected and then its power is turned on, rightly set the type of the encoder, the pulse output direction along each axis and the axis movement diretion, etc.; refer to the relevant setting steps：

① Correctly set the bit 5 of parameter No.1815 (APCx) to select whether the servo is used the absolute encoder or not based upon the encoder types of the servo motor.

② Correctly set the gear ratio CMR/DMR (Set by parameters No. 1816 and No. 1820) based upon the driving ratio of the machine tool, so that the movement distance of the machine tool is consistent with the program command value. (Refer to the consecutive chapters)

③ If the machine tool movement direction is inconsistent with the required of the shifting command, the bit 2 of parameter No.1811 (PODx) can be set to reverse the movement direction output by the corresponding servo axis command.

Note 1: In the Auto or MDI mode, when the direction of the command movement axis is negative to the feed direction of the actual axis, modify the axis corresponding by bit 2 of parameter 1811.

Note 2: In the Manual mode, when the manual movement direction is negative to the feed direction of the actual axis (The axis movement direction is correct in the Auto or MDI mode), alter the corresponding PLC data parameters K8.0～K8.4 of this axis.

Relevant Parameter			
	Para. No.	**Bit**	**Parameter Meaning**
988TA series system parameter	**1811**	#2	Pulse output direction selection along each axis 0: Positive 1: Negative
	1815	#5	Position detector 0: Without using an absolute position detector 1: Use an absolute position detector
PLC data parameter	**K8.0～K8.4**		Whether the manual movement direction along each axis is negative. 0: Positive 1: Negative

Table 2-8

2.6 Acceleration/Deceleration Character Adjustment

The more the acceleration/deceleration time constant is, the slower the acceleration/deceleration procedure is, the less the impact of machine movement is, the slower the maching efficiency is; The less the acceleration/deceleration time constant is, the faster the acceleration/deceleration procedure is, the more the impact of machine movement is, the higher the maching efficiency is.

When the acceleration/deceleration time constant is same, the higher the start/end speed of the acceleration/deceleration is, the faster the acceleration/deceleration procedure, the stronger the impact of machine tool movement is, the higher the machining efficiency is; the lower the start/end speed of the acceleration/deceleration is, the slower the acceleration/deceleration procedure, the weaker the impact of machine tool movement is, the slower the machining efficiency is

The principle of the acceleration/deceleration character is adequately reduced the acceleration/deceleration time constant and improved the start/end speed of the acceleration/deceleration to enchance the machining efficiency under the premise of the no alarm on drive, no step-out on motor and no impact on machine tool movement.

Note: ◇ When bit 4 of parameter 1601 equals to 0, the feedrate should be reduced to the start speed of the deceleration at the path intersection of the cutting feed, and then accelerate to the command speed to the adjacent blocks. The machining efficency may reduce when the accuracy positioning may carry out at the intersection point of the path.

◇ When bit 4 of parameter 1601 equals to 1, the adjacent cutting path is directly performed the smooth transition based upon the acceleration/deceleration method, the feedrate may not reduce to the start speed when the previous path is ended instead of forming an arc transition (Non-accuracy positioning) at the intersection point of the path; And therefore, the workpiece surface is smooth and the machining efficiency is higher based upon the path transition method.

Relevant Parameter		
Para. No.	**Bit**	**Parameter Meaning**
1420		Rapid traverse rate along each axis
1421		The lowest speed (F0) of rapid traverse override along each axis
1422		The top cutting feedrate of overall axes
1423		Manual feedrate along each axis
1424		Manual rapid traverse rate along each axis

1425		The FL speed of reference position return along each axis
1428		Reference position return speed of each axis
1434		The Max. feedrate of the manual MPG feed of each axis
1466		The feedrate in tail-retraction of the execution of the thread cutting
1601	#4	When the block is performed at the rapid traverse rate: 0: Not overlapping (Accuracy in-position) 1: Overlapping (Smooth transtion)
1610	#4	The acceleration/deceleration of the manual feed: 0: Exponential acceleration/deceleration 1: Linear acceleration/deceleration after interpolation
1620		The linear acceleration/deceleration time constant T of each axis
1622		The acceleration/deceleration time constant of cutting feed after interpolation along each axis
1624		The acceleration/deceleration time constant with manual feed of each axis after the interpolation
1625		The FL speed of the exponential acceleration/deceleration with manual feed of each axis
1626		The exponential acceleration/deceleration time constant during the thread cutting cycle of each axis.
1627		The FL speed of the exponential acceleration/deceleration time constant during the thread cutting cycle of each axis.
1628		The acceleration/deceleration time constant of the tail-retraction operation during the thread cutting cycle of each axis.

Table 2-9

2.7 Reference Point & Softlimit

The system supports 3-kind of mechanical zero establishment (It is also called the reference point setting) methods of the GSK988TA/988TA1/988TB series: no stopper reference point setting, stopper reference point setting and absolute encoder reference point setting.

Reference setting method	System parameter setting
Absolute encoder reference point setting	Bit 5 of parameter 1815 (APCx) sets to 1.
No stopper reference point setting	Bit 5 of parameter 1815 (APCx) sets to 0. Bit 1 of parameter 1002 (DLZ) sets to 1 or Bit 1 of parameter 1005 (DLZx) sets to 1 (Any of them can be set to 1)
Stopper reference point setting	Bit 5 of parameter 1815 (APCx) sets to 0. Bit 1 of parameter 1002 sets to 0 and bit 1 of parameter 1005 DLZx sets to 0.

Table 2-10

Note 1: When the absolute encoder is used and the reference point is set up, the system may automatically save the reference point position after the power is turned off; the reference point position should not set again when the machining is performed with the power-on next time.

Note 2: The reference point setting operation should be performed to set up the reference point when the system power is turned on by using the non-stopper/block reference point setting.

Relevant Parameter			
	Para. No.	**Bit**	**Parameter Meaning**
GSK988TA/988TA1/988TB series system parameter	**1005**	#0	When the reference point does not establish in the Auto operation (MEM, DNC or MDI); Whether the system alarm occurs when specifying the movement command other than the G28. 0: With alarm 1: Without alarm
	1006	#5	Set the reference position direction return of each axis 0: Positive 1: Negative
	1201	#2	The local coordinate system after the manual reference point return. 0: Not cancel 1: Cancel
	1240		Set the coordinate value of the 1st reference point in the mechanical coordinate system.
	1241		Set the coordinate value of the 2nd reference point in the mechanical coordinate system.
	1242		Set the coordinate value of the 3rd reference point in the mechanical coordinate system.
	1243		Set the coordinate value of the 4th reference point in the mechanical coordinate system.
	1425		Set each axis's speed (FL speed) after decelerating when the reference position return is performed.
PLC data paramete	**K12.2**		Whether the zero return operation direction button is performed the self-locking 0: Without self-locking 1: Self-locking

Table 2-11

2.7.1 Reference Point Setting of Absolute Encoder

When the machine tool configures the absolute encoder and the system does not establish the reference point yet, the reference position return operation of the absolute encoder should be performed. After the tool returns to the reference point, the end LED of the reference position return lights up, and then the coordinate system setting is automatically executed.

The operation methods of the absolute encoder reference positon return：

① In the Manual 

or MPG 

mode, move the machine to the reference point position to be set.

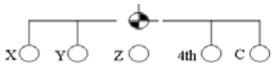
② Set the bit 4 of parameter 1815 (APZx) to 0, the system alarm occurs; the power is then turned on again after its power is cut off.

③ Control the reference point return switch 

.

④ Press the corresponding feed axes 

of the reference position return, and then perform reference point return operation.

⑤ The reference point return completion lights (LED) 

of its corresponding axes are lighten up, then reference position return ends; in this case, the system registers the current encoder position and automatically set the bit 4 of parameter 1815 (APZx) to.

Relevant Parameter		
Parameter No.	**Bit**	**Parameter Meaning**
1815	**#4**	The detector position between mechanical position and absolute one when using the absolute encoder. 0: Consistent 1: Inconsistent
1815	**#5**	Position detector 0: Regardless of the absolute position detector 1: Use the absolute position detector

Table 2-12

2.7.2 Block Reference Position Setting

When the system sets the non-stopper reference positon return to disabled (that is, the block reference point setting is enabled), the machine tool should be installed the deceleration switch to realize the function of reference position return.

The operation methods of the block reference position return:

① Confirm the overtravel limit switch is enabled.

② Press the reference return button 

to shift the system working method to reference position turn.

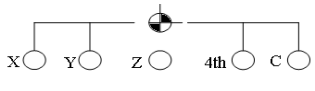
③ In order to reduce the zero return speed, adjust the rapid traverse override switch 

to the low speed shift.

④ Press the direction selection switch of the corresponding feed axis for the reference position return, the system reads the corresponding axis and direction selection signal, and then return to the reference position operation.

⑤ The planker moves to the deceleration point at the rapid traverse rate, the deceleration signal DECx is enabled (The valid LEVEL of the signal is set by bit 5 of parameter 3009 (DECx)), it consecutively moves to reference point by zero return low speed set in parameter 1425 after decelerating.

⑥ Tool leaves from the deceleration switch position; the system begins to detect the one-turn signal nPC of motor after the deceleration signal DECx changes.

⑦ After the system detects the 1st nPC signal of the motor, the reference point return end signal ZPx and the establishment signal ZRFx are set to 1; the indicator (LED) 

of the reference position return lights up, the reference position return ends accordingly.

Note: Generally, the mechanical zero block installs at the Max. stroke position, and the zero return block efficient stroke is above 25mm; in order to ensure an adequate deceleration distance for reducing the speed, the accuracy zero return can be performed accordingly. The faster the mechanical zero turn is, the longer the zero return blocker is; otherwise, the movement planker will push through the zero return blocker due to CNC acceleration/deceleration, machine tool inertia, etc.without an adequate deceleration distance, so that the zero return accuracy wil be affected. In addition, ensure that the planker will not be interfered to the other components with the machine tool during the zero return movement of the planker; ensure that it is safe.

Usually, the connection method for matching the AC servo motor: separately use one stroke switch and one-turn signal of servo motor; refer to the following figure:

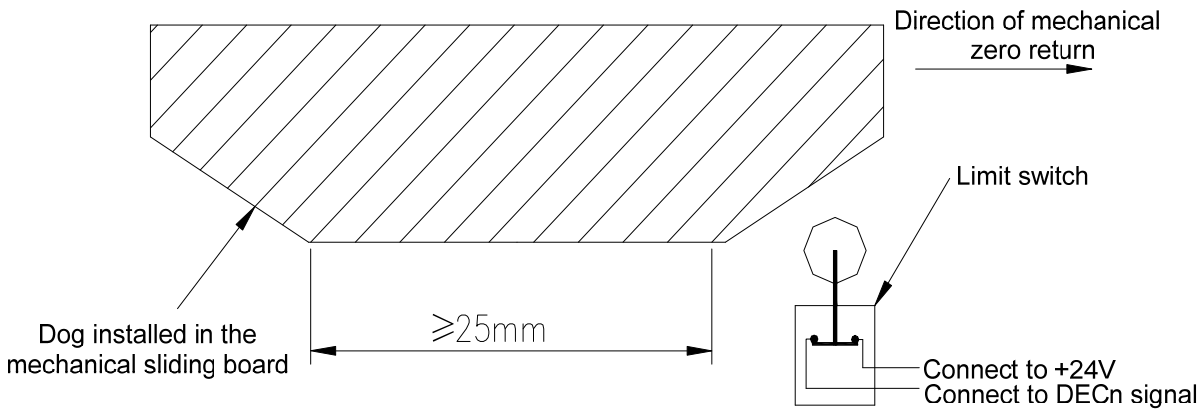


Fig. 2-2

It is necessary to avoid the boundary point position of encoder one-turn signal after the stroke switch is released and when the mechanical zero return is performed and after the deceleration switch is delivered by using this connection method. Guarantee that the motor reaches to the one-turn signal of encoder after revolving half circle for enhancing the zero return accuracy. Slightly adjust the block position to reduce the zero return error.

2.7.3 Non-Stopper Reference Point Setting

When the function of the non-stopper reference position return is set to enable by system, the machine tool can be returned to the reference position instead of installing the deceleration switch. After the tool returns to the reference position, its LED lights up and the coordinate system is automatically set accordingly.

The operation method of reference point return without block:

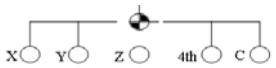
① The machine tool performs the feed along axis based upon the reference position return, and then stop at neighbour but lower of the reference point.

② Control the reference point return switch 

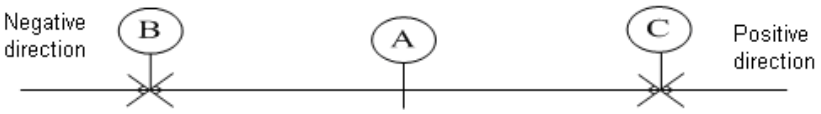
.

③ Press the direction selection switch of the corresponding feed axis for the reference position return, the corresponding axis and direction selection signal Jx sets to 1, and then start returning the operation of the reference position.

④ Tool moves to the reference point based upon the directin set by bit 5 of parameter 1006 (ZMIx) and the FL speed set in parameter 1425.

⑤ After the system detects the 1st PC signal of the motor, the reference point return end signal ZPx and the establishment signal ZRFx are set to 1; the indicator (LED) 

of the reference position return lights up, the reference position return ends accordingly.



A：Perform the position before the non-stopper reference point return

B：Perform the reference point position after the reference point returns negatively; that is, the 1st PC signal generates where the point A is moved negatively.

C：Perform the reference point position after the reference point returns positively; that is, the 1st PC signal generates where the point A is moved positively.

Relevant Parameter		
Parameter No.	**Bit**	**Parameter Meaning**
1002	**#1**	Whether the non-block reference point setting function is enabled: 0: Disabled 1: Enalbed (The overall axes are enabled)
1002	**#3**	G28 command when the reference point does not establish: 0: Similar as the manual reference point return, perform the reference point return by using the deceleration block. 1: P/S alarm occurs.
1005	**#1**	Whether the non-block reference point setting function is enabled: 0: Disabled

		1: Enalbed
1300	**#6**	From the LZR power-on to the manual reference point return, whether perform the 1st stored stroke detection. 0: Not performed 1: Performed

Table 2-13

2.7.4 Stored Stroke Dection Setting

There are 3 stored stroke detection areas are provided in the system of the GSK988TA/988TA1/988TB series: Stored stroke limit detection 1, 2 and 3; and tool can not be entered the 3 specified areas.

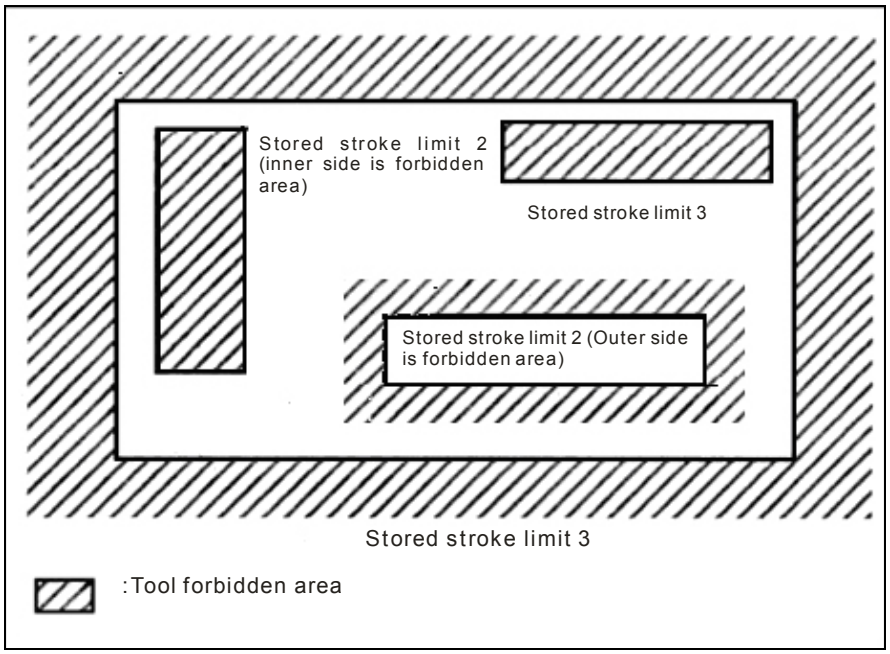


Fig. 2-3

Stored stroke limit detection 1:

The boundary is set by parameter 1320, 1321 or 1326, 1327; the outside of the setting boundary is forbidden area. Usually, the manchine tool manufacture sets this area as the Max. stroke range.

Note 1: The bit 7 of parameter 1300 is only enabled to the stroke limit detection 1.

Note 2: When the parameter #1300.7 equals to 1 and when the "program" is executed, judge whether the path exceeds the stored stroke after the block is performed in advance before carrying out the current block. If does, the alarm of the overtravel occurs; otherwise, this block will be consecutively performed.

Stored stroke limit detection 2 (G22　G23)：

The boundary is set by parameter 1322, 1323 or command, set whether the forbidden area is inside or outside of the setting boundary is determined by bit of 0 of parameter 1300. Prohibit the tool entering the the forbidden area by G22 command is used in programming, and the G23 commands allows that the tool enters to the forbidden area; the G22 and G23 in the program should be separately specified, which are the individual blocks. Refer to the G command for details.

Stored stroke limit detection 3:

The boundary is determined by parameter 1324, 1325; the inner side of the setting 3 boundary is forbidden area.

Note: If two points are set as the absolute same in the setting forbidden area, the area is shown below:

1. When the stored storke detection 1 is in the setting forbidden area, if two points are equal, the overall areas are regarded as the forbidden areas.

2. When the stored storke detection 2 or 3 is in the setting forbidden area, if two points are equal, the overall areas are treated as the movable areas.

Display the alarm time:

The bit 7 of parameter 1300 (BFA) selection is whether immediately displayed the alarm before or after entering the forbidden area.

Release the overtravel alarm:

When the tool exceeds the stored stroke limit, the alarm shows and the tool decelerates then stops; shift to the manual mode to move the tool out of the forbidden area negatively (It moves out negatively if it is performed the overtravel positively; and it moves out positively if it is performed the overtravel negatively); clear the alarm display by pressing the resetting button.

Relevant Parameter		
Para. No.	**Bit**	**Parameter Meaning**
1300	**#0**	The set forbidden area of the stored stroke detection 2 by parameter (No.1322，No1323) is: 0: Internal area　　1: External area
1300	**#2**	Whether the stored stroke detection shifting signal EXLM is enabled: 0: Disabled　　1: Enabled
1300	**#5**	Whether the releasing signal RLSOT3 of the stroke detection 3 is

		enabled: 0: Disabled 1: Enabled
1300	**#6**	Whether perform the 1st stored stroke detection after the power is turned off till to the reference position return. 0: Executed 1: Not executed
1300	**#7**	When the issue exceeds the command of the stored stroke: 0: Alarm occurs after exceeding the stroke 1: Alarm occurs before exceeding the stroke
1310	**#0**	Whether perform the stored stroke detection 2 of each axis: 0: Not executed 1: Executed
1310	**#1**	Whether perform the stored stroke detection 3 of each axis: 0: Not executed 1: Executed
1320		The coordinate value (PC1) of stored stroke detection 1 of each axis of the positive direction boundary
1321		The coordinate value (NC1) of stored stroke detection 1 of each axis of the negative direction boundary
1322		The coordinate value (PC2) of stored stroke detection 2 of each axis of the positive direction boundary
1323		The coordinate value (NC2) of stored stroke detection 2 of each axis of the negative direction boundary
1324		The coordinate value (PC3) of stored stroke detection 3 of each axis of the positive direction boundary
1325		The coordinate value (NC3) of stored stroke detection 3 of each axis of the negative direction boundary
1326		The coordinate value II (PC12) of stored stroke detection 1 of each axis of the positive direction boundary
1327		The coordinate value II (NC12) of stored stroke detection 1 of each axis of the negative direction boundary

Table 2-14

2.8 Stored Pitch Error Compensation

If the pitch error compensation data is defined, the pitch error compensation along each axis can be performed based upon the detection unit of each axis.

Set the pitch error compensation data for each compensation position, and its compensation position is set based upon the defined interval of each axis. The compensation origin is the reference

position of the tool return.

The following parameters should be set when the pitch error compensation is performed:

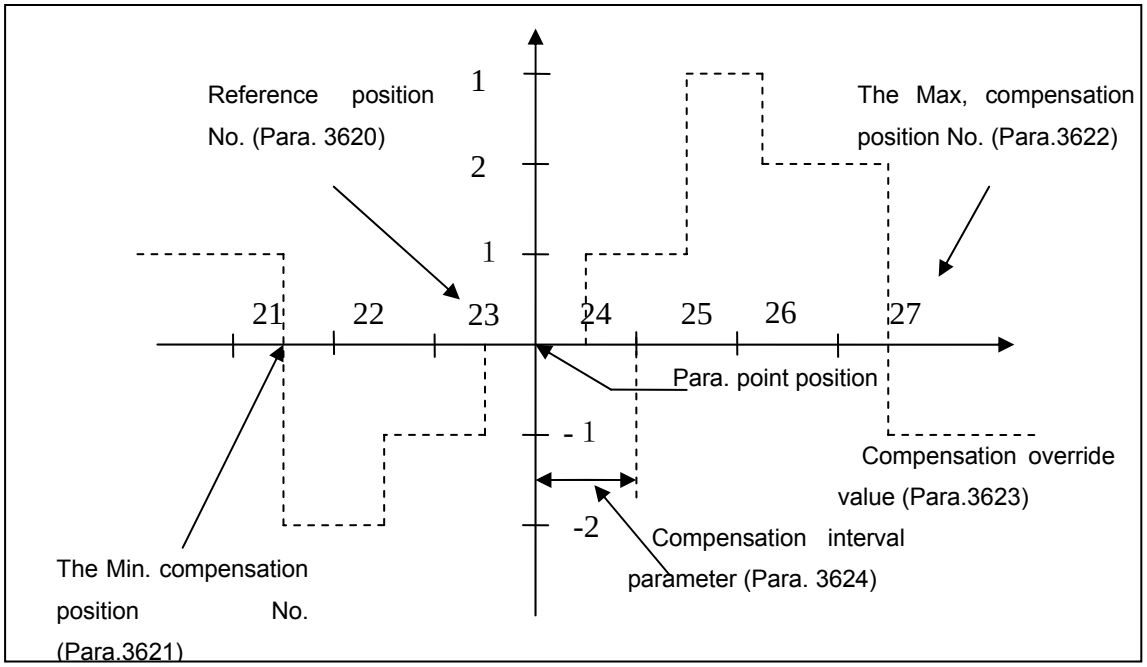
Parameter 3620: Each axis locates on position number of pitch error compensation of the reference point.

Parameter 3621: The least position number of pitch error compensation of each axis.

Parameter 3622: The top position number of pitch error compensation of each axis.

Parameter 3623: The override of pitch error compensation of each axis.

Parameter 3624: The position interval of pitch error compensation of each axis.



Compensation number	21	22	23	24	25	26	27
Compensation value of setting	-3	+1	+1	+1	+2	-1	-3

Compensation position definition:

In order to specify the compensation value of each axis, specify the positive/negative movement direction of compensation based upon the reference point. If the machine tool stroke exceeds the specified range on the positive/negative direction, in that way, the pitch error compensation does not work out its range.

Compensation position number:

On the setting page of the pitch error compensation, there are 1024 (0 ～1023) compensation

positions can be used. The position numbers can be freely distributed of each axis by parameter. The compensation position number (Parameter 3620) of reference point, the least position number (Parameter 3621) of compensation and the maximum position number (Parameter 3622) of compensation of each axis should be set.

For example:

I. Linear axis

Machine tool stroke: －400mm~＋800mm

Position interval of pitch error compensation: 50mm

Compensation position number of reference point: 70

After the above-mentioned definitions are performed, the furthest compensation position number along the negative direction is shown below:

The compensation position numer - of reference point (the machine tool stroke/compensation position interval along negative direction) ＝70－400/50+1=63

The furthest compensation position number along with positive diection is shown below:

The compensation position numer + of reference point (the machine tool stroke/compensation negative interval along negative direction) ＝70＋800/50=86

The corresponding relationships between machine tool and compensation point position are shown below:

Parameter	Setting value
3620: Reference position compensation number	70
3621: The minimum compensation position number	63
3622: The maximum compensation position number	86
3623: Compensation override	1
3624: Pitch error compensation position interval	50000

II. Rotation axis

Movement value per revolving: 360 degree

Pitch error compensation position interval: 45 degree

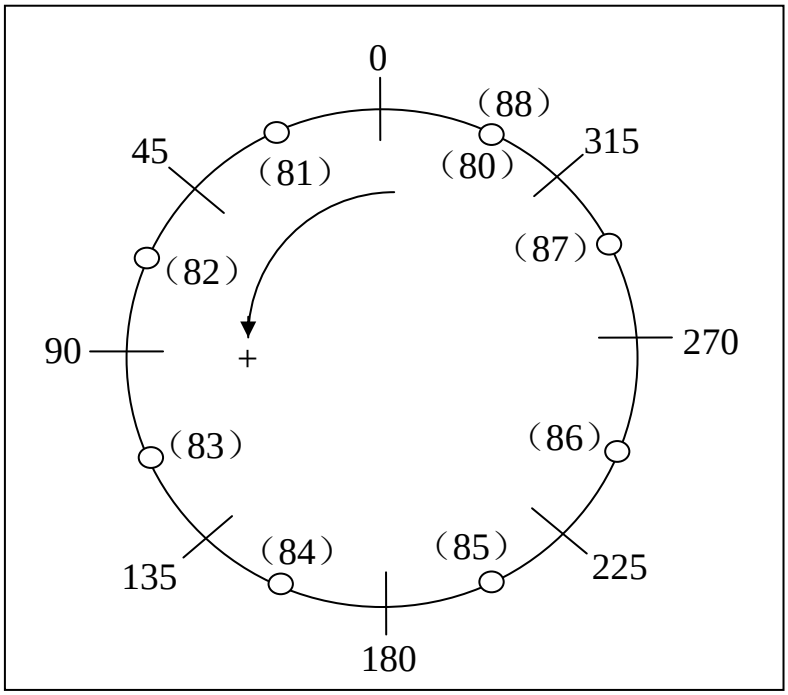
Compensation number of reference position: 80

After the above-mentioned parameters are defined, the furthest compensation position number of the rotation axis along negative direction equals to compensation position number of the reference point.

The furthest compensation position number along with positive diection is shown below:

The compensation position number of reference point + (movement value per revolving/compensation position interval) =80+360/45=88

The corresponding relationships between the the machine tool and compensation position number are shown below:



Parameter setting is as follows:

Parameter	Setting value
3620: Reference position compensation number	80
3621: The minimum compensation position number	80
3622: The maximum compensation position number	88
3623: Compensation override	1
3624: Pitch error compensation position interval	45000

If the sum of the compensation values from positions 81~88 are not regarded as 0, the position error may occur. The so-called sum is the accumulation of the pitch error compensation value of each rotation. Additionally, the identical compensation values should be set at the 80 and 88 compensation positions.

For example:

Compensation position number	80	81	82	83	84	85	86	87	88
Compensation value of setting	+1	-2	+1	+3	-1	-1	-3	+2	+1

Pitch error compensation value setting

In the sytem page set, enter the pitch compensation page by PITERROR, refer to the following figure:

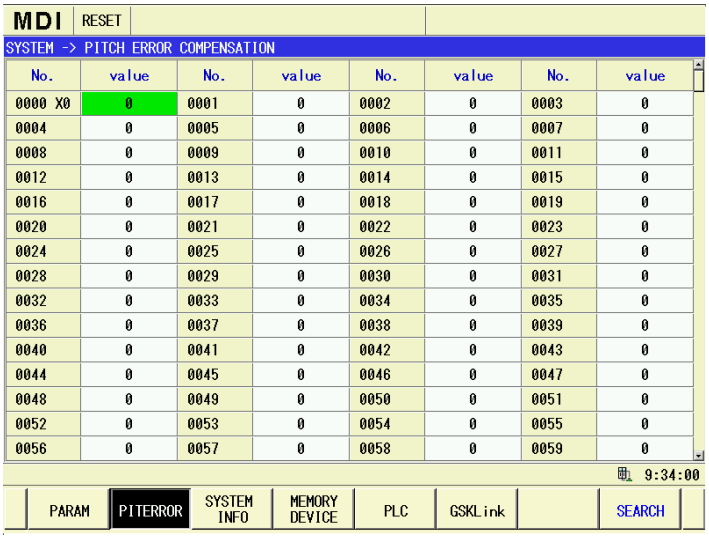


Fig.2-4

① In this page, user can view and set the pitch compensation value corresponding by each pitch number.

② In the pitch compensation page, select the desired compensation value of the pitch number to be set by page button or and cursor movement key , , or ; search the pitch number by SEARCH softkey, and then position the cursor at the desired pitch number compensation number to be modified.

③ The selected compensation number compensation value by INPUT is on the revisable state, input the compensation value by numerical value buttons, and then complete the modification by INPUT.

Icon Explanation	
0010 X0, 0030 Z0	The compensation number sety by parameter 3620 corresponding axis

0001 X-, 0020 Z-	The compensation number sety by parameter 3621 corresponding axis
0014 X+, 0034 Z+	The compensation number sety by parameter 3622 corresponding axis

Note: The interval of the pitch error compensation value or point is related with the axis of which it is diameter programming or radius programming (The diameter/radius programming is directly affected the least movement unit of the system). If the axis is diameter programming, the parameter setting value is a diameter; if it is the radius programming, and the setting value is a radius, which are all regarded as the detection units.

Relevant Parameter		
Para. No.	**Bit**	**Parameter Meaning**
3620		The pitch error compensation number of reference point along each axis
3621		The furthest pitch error compensation point number of each axis along negative direction
3622		The furthest pitch error compensation point number of each axis along positive direction
3623		Pitch error compensation override along each axis
3624		The distance of pitch error compensation point along each axis
3628		The setting value of pitch compensation pulse frequency

Table 2-15

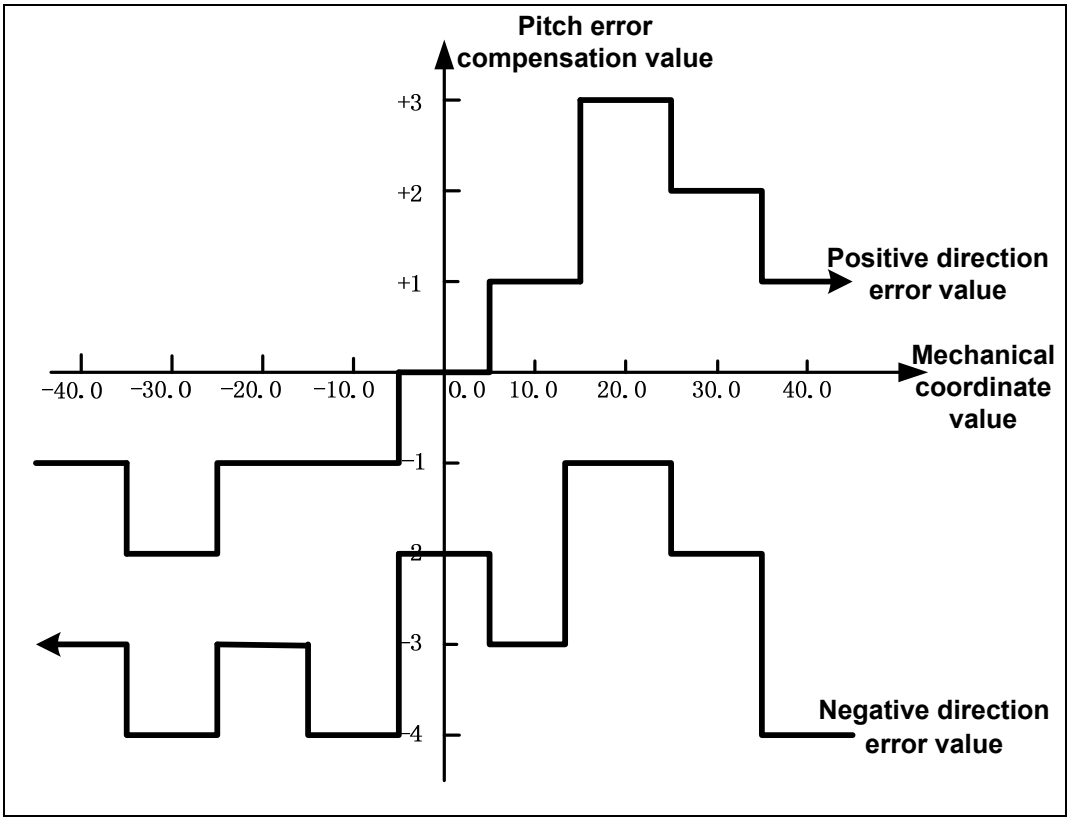
2.9 Bi-Directional Pitch Error Compensation

The stored pitch error compensation does not aim to the distinguishment of the movement direction, however, in the bi-directional pitch error compensation; it can be differentially set the pitch error compensation value along the negative and position direction to perform the pitch error compensation with different directions. What's more, when the movement rotates negatively, the compensation value can be automatically calculated based upon the pitch error compensation data to perform the compensation of the negative movement direction, as the negative interval compensation. Thus, further reduce the path error of the positive and negative directions.

Set the bi-directional pitch error function to enable (Parameter №3605#0="1") if you want to use this function, refer to the following settings:

Parameter	Content

No.	
3605#0	Whether use the bi-directinal pitch error compensation 0: Do not use 1: Use
3620	The pitch error compensation number of the reference point along each axis
3621	The number of the farthest pitch error compensation point along the negative direction of each axis
3622	The number of the farthest pitch error compensation point along the positive direction of each axis
3623	Set the override of pitch error compensation along each axis
3624	The interval of pitch error compensation point along each axis
3626	The the compensation point number of the closest negative side of the bi-directional pitch error compensation
3627	The pitch error compensation value in the reference point when moving to this point from the opposite direction to the origin return



Pitch error compensation data along positive direction

Compensation point No.	20	21	22	23	24	25	26	27
Compensation	-1	+1	0	+1	+1	+2	-1	-1

value setting								

Pitch error compensation data along negative direction

Compensation point No.	30	31	32	33	34	35	36	37
Compensation value setting	-1	+1	-1	+2	-1	+2	-1	-2

Throughtout set the data value observed from the negative side of the coordinate system when the pitch error compensation data is set. The pitch error data along the negative direction should be identical with the setting point by the positive pitch error compensation data. The pitch error data along negative direction is also set the viewed data value from the negative side of the corrdinate system.

Parameter No.	Setting value	Content
3605#0	1	Whether use the bi-directional pitch error compensation 0: Do not use 1: Use
3620	23	The pitch error compensation number of reference point along each axis
3621	20	The number of the furthest pitch error compensation point along negative direction of each axis
3622	27	The number of the furthest pitch error compensation point along positive direction of each axis
3623	1	Set the compensation override of the pitch error along each axis
3624	100000	The interval of pitch error compensation point along each axis
3626	30	The compensation point nearest to the negative side for the bi-directional pitch error compensation
3627	-2	The pitch error compensation value in the reference point when moving to this point from the opposite direction to the origin return

2.10 Reverse Interval Compensation

Machine tool may lose a part of movement values when performing the reverse movement negatively, due to the error generates on the driving mechanism, so that the machining accuracy is then affected. In order to reduce the error causing from reverse movement, the system provides reverse interval error compensation function.

The axis and its reverse interval compensation value are related with the diameter or radius programming (The diameter/radius programming is directly affected to the system least movement unit). If the axis is diameter programming, the parameter setting value is diameter; if it is the radius programming, the setting value is radius accordingly.

$$Detectio\ nunit = \frac{The\ least\ movement\ unit}{Command\ multiplication\ ratio\ (CMR)}$$

The reverse interval compensation can be improved the machining accuracy after it compensates precisely, which can be measured by dialgauge, micrometer dialgauge or laser detector. In order to perform the accurately compensation, it is not recommended to use the MPG or single-step method for measuring the lead screw interval interval instead of measuring by the following interval interval:

Editing program:

O0001；

N10 G01 W10 F800 ；

N20 W15 ；

N30 W1 ；

N40 W-1 ；

N50 M30

① The reverse interval error compensation value should be set to before measuring;

② The program is prformed with single block, it finds the measurement reference point 1 after positioning twice, record the current data, and then operate 1mm along its same direction, lastly, reversely operate 1mm to point 2, read the current data accordingly.

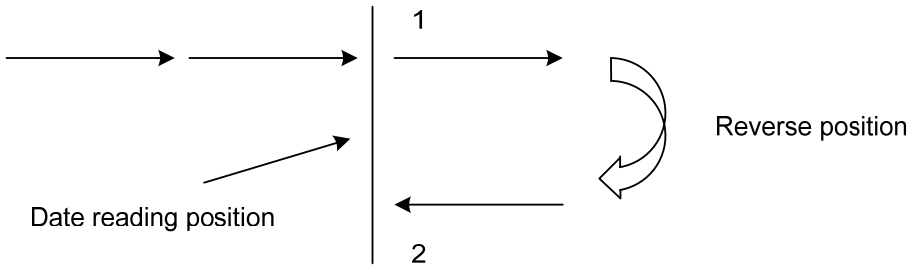


Fig. 2-9-1 The schematic of reverse interval measurement method

③ Reverse interval error compensation value = |The recorded data form point 1 –The recorded data form point 2|; the calculated data is inputted to the parameter №1851 in CNC data, after it is

converted into detection unit.

Data 1: Read to the dialgauge data from 1;

Data 2: Read to the dialgauge data from 2;

Detection unit = The least movement unit/ CMR;

For example: The system sets as IS-C by parameter (Bit 1 of parameter No. 1004 sets to 1), in the metric system machine tool (Parameter No.1001#0 INM sets to 0), if the setting value of parameter No. 1820 (It is used for setting the command multiplication ratio of each axis) is 1, therefore, the command multiplication ratio CMR of the system equals to 1;

So: X axis: Detection unit = The least movement unit/ CMR＝0.00005mm / 1 ＝0.00005 mm;

Z axis: Detection unit = The least movement unit/ CMR＝0.0001mm / 1 ＝0.0001 mm

If the reverse interval error compensation value of X axis measured by dialgauge is 0.0150mm, set the parameter No.1851 to 300; the reverse interval error compensation value along Z axis measured by dialgauge is 0.0300mm, and then set the parameter No.1851 to 300.

The steps of parameter setting of reverse interval are shown below:

① Measure the reverse interval compensation value based upon the above-mentioned methods, and then save to the parameter No.1851; note that the parameter unit is detection one.

② After the reverse interval compensation value is set, set its output method based upon the bit 7 of parameter No.1800 (BDEC); 0: Fixed pulse frequency output 1: It outputs according to the deceleration character.

③ When bit 7 of parameter No.1822 (BDEC) sets to 0 (Fixed pulse frequency output), set its pulse output frequency by parameter 1800#6 (BD8); 0: Compensation with the set frequency 1: 1/8 compensation with the set frequency. The setting frequency of the compensation is performed in the parameter No.1853.

④ When bit 7 of parameter No.1800 (BDEC) sets to 1 (It outputs based upon acceleration or deceleration character), the enabled time constant of acceleration/deceleration can be set by parameter 2071.

Relevant Parameter		
Parameter No.	**Bit**	**Parameter Meaning**
1800	**#6**	Pulse output frequency of reverse interval compensation 0: The compensation can be performed based upon the frequency set by parameter #1852. 1: The 1/8 compensation can be performed based upon the frequency set by parameter #1852.

1800	**#7**	Reverse interval compensation method 0: Fixed pulse frequency out (It is set by parameters ＃1853 and #1800.6) 1: Pulse frequency outputs according to an acceleration or a deceleration character.
1851		Reverse interval compensation value along each axis
1852		Reverse interval compensation value at the rapid traverse rate of each axis
1853		The setting value of pulse frequency of reverse interval compensation
2071		The enabled time constant of reverse interval acceleration/deceleration of each axis.

Table 2-16

2.11 Spindle Function Adjustment

2.11.1 Spindle Encoder

GSK988TA/988TA1/988TB series owns two-circuit encoder input interface (CN21, CN22); the GSKLink communication spindle feedback value is used, which regards as the feedback input of spindle speed in the default condition. It is necessary to set the relevant parameter and signal of each spindle encoder for correctly read the spindle actual speed during using.

Relevant Parameter			
Parametr No.	**Bit**	**Parameter Meaning**	**Remark**
3720		Encoder linear number of each axis	Set the encoder linear number (1~99999999) of each spindle
3721		Gear number at one side of the gear of each spindle position encoder	This parameter is used for setting the gear number ratio (setting range: 1~9999) during the speed control.
3722		Gear number at one side of each spindle	

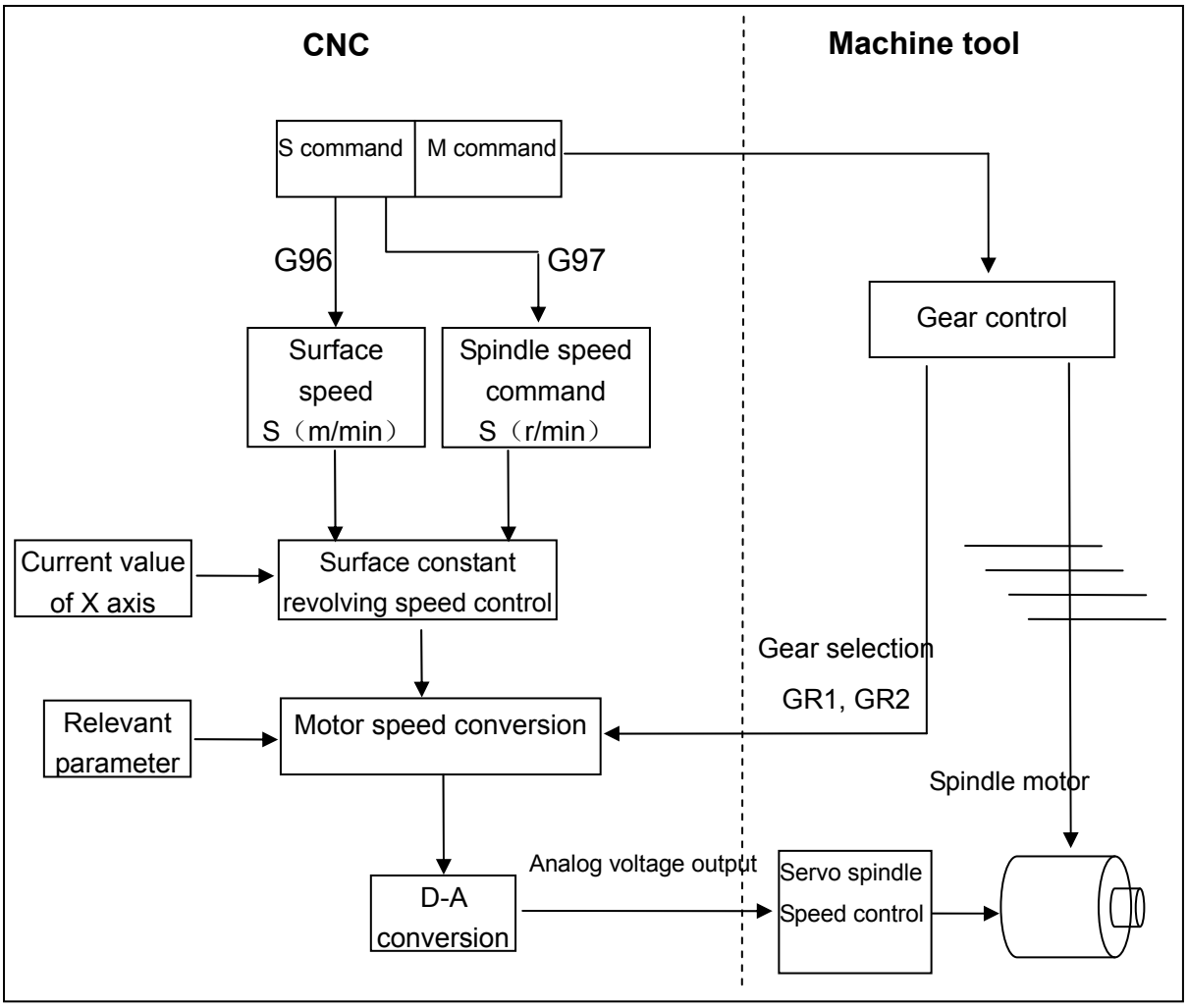
Table 2-17

2.11.2 Spindle Speed Control

There are 2 methods for spindle speed control, which can be selected by parameter. The first one is that servo drive is controlled by adopting the communication method when connecting with GSKLink spindle. The second one is that the 0～10V analog voltage is delivered to the spindle servo

drive device or converter by using analog voltage output port of the extension I/O unit. The infinite variable speed can be carried out in despite of using what kind of method.

Although the S command is spindle speed, the actual controllable object is spindle motor. Therefore, the speed and gear-level of spindle motor from CNC should have a relevant corresponding relationship. This system confirms the current used gears of the machine tool by the gear selection signal (GR1, GR2); CNC outputs the corresponding spindle speed with the gear. The spindle speed control procedure is as follows:



When the speed specified by programming is inconsistent with the actual spindle speed, adjust the specified speed consistent with the actual one by debugging the data parameter №.3730, №.3731. Its adjustment divides into two methods based upon whether is connected with the spindle encoder.

1．Unused spindle encoder:

① The parameters 3730 and 3731 should be separately set to 1000 (The gain adjustment data of

spindle speed analog output) and 0 (The compensation value of the offset voltage for the spindle speed analog speed) before debugging; cut off the connection between the CNC and spindle after the power is turned off, and then turn the power again, then perform the a M (M41—M44) code of common-use spindle gear (The system is regarded as the 1st gear by default after its power is turned on);

② In the MDI mode, specify the top speed S code of the described gear, for example, after the 1st gear is selected, input the Max. speed command (Parameter 3741) in the MDI page and then put the cycle start button;

③ Measure the output voltage SVC based upon the provided interface definition from the above-mentioned descriptions.

④ Set the following formula's value in the parameter 3730:

$$\text{setting value} = \frac{10\text{V}}{\text{Measured voltage (V)}} \times 1000$$

⑤ Specifying the spindle 1st gear speed analog output is the spindle speed (parameter 3741) with the most voltage again, after the parameter is set, and confirm the output voltage is 10volt.

⑥ Specify S0 in MDI mode;

⑦ Measure the output voltage SVC;

⑧ Set the following formula's value in the parameter 3731:

$$\text{setting value} = \frac{-8191 \times \text{Offset voltage (V)}}{12.5}$$

⑨ Specify S0 again after setting the parameter, and then confirm that the voltage is 0V.

2. Use the spindle encoder

① The parameters 3730 and 3731 should be separately set to 1000 (The gain adjustment data of spindle speed analog output) and 0 (The compensation value of the offset voltage for the spindle speed analog speed) before debugging; After correctly connecting and setting the spindle encoder, then perform the a M (M41—M44) code of common-use spindle gear (The system is regarded as the 1st gear by default after its power is turned on);

② In the MDI mode, specify the top speed S code of the described gear, for example, after the 1st gear is selected, input the Max. speed command (Parameter 3741) in the MDI page and then put the cycle start button to rotate the spindle;

③ Record the actual speed value in the position page, in this case, the actual value should be similar with the specified one; if there are big differences between them, it is better to check whether the encoder parameter is correct.

④ Set the following formula's value in the parameter 3730:

$$\text{setting value} = \frac{\text{Setting value of parameter No. 3741}}{\text{Actual rotation speed}} \times 1000$$

⑤ Specifying the spindle 1st gear speed analog output is the spindle speed (parameter 3741) with the most voltage again, after the parameter is set, and confirm the actual speed is the setting value in parameter 3741.

⑥ Specify S0 in MDI mode;

⑦ Record the actual speed value in the position page;

⑧ Input the recorded actual speed in parameter 3731;

⑨ Specify S0 again after the parameter is set, and then confirm that the output speed is 0.

Relevant Parameter			
Para. No.	**Bit**	**Parameter Meaning**	**Remark**
3031		Set the allowance digit of the S code	
3708	**#0**	Whether check the spindle speed arrival signal 0: YES 1: NO	
3708	**#1**	Whether check the spindle speed arrival signal when performing the thread cutting. 0: It is set by parameter SAR 1: Detect	
3708	**#6**	When the thread is performed the machining or tapping cycle, the spindle override is shown below: 0: Disabled (Fix at 100%) 1: Enabled	
3710		CNC control spindle number	Set the spindle numbers (1~3) controlled by CNC
3713	**#6**	When the program command based upon the address P is selected the spindle in the multi-spindle control, whether automatically performs position encoder feedback shifting in the thread cutting/feed per rev. 0: NO 1: YES	
3717		Amplifier number of each axis	The spindle sets to 1～99 (It is set consistent with the spindle drive) by connecting with the GSKLink

			The spindle sets to -1~-2 (correspond to the remote I/O unit 1, spindle interface 1 and 2), -3~-4 (correspond to the remote I/O unit 2, spindle interface 1 and 2)
3720		Encoder linear number of each spindle	Set the encoder linear number (1~9999) of each spindle
3721		Gear number at one side of position encoder of each axis	The two parameters are used to set the gear number during the speed control. (Setting range: 1~9999)
3722		Gear number at one side of each spindle	
3723		The corresponding channel number (0~4) of each spindle encoder	It rotates by GSKLink feedback when setting to 0. Set to the 1~4, the corresponding encoder interface 1~4
3730		Gain adjustment data of each spindle speed analog output	Setting value range: 700~1250, data unit 0.1%
3731		The compensation value of offset voltage for each spindle speed analog output	Setting value range: -1024~1024
3740		Check the delay time of the spindle speed arrival signal	
3741		The top spindle speed of gear 1	Set the speed of corresponding gear of each spindle (0~32767 r/min)
3742		The top spindle speed of gear 2	
3743		The top spindle speed of gear 3	
3744		The top spindle speed of gear 4	
3770		The axis is regarded as the calculation reference during the constant linear speed control.	The axis (0,1 ～ controllable axis number) is regarded as the calculation reference when the constant linear speed is controlled, the X axis is treated as reference when setting to 0.
3771		The lowest speed of spindle during the constant linear control	The lowest speed when setting G96 (0~32767 r/min)
3772		The upper-limit speed (0~32767 r/min) of each axis	
3775		Default spindle selection P command value (MPD) in the multi-spindle	Set the default P command value without specifying S_ P_ at all after the power is turned on.
3781		Select the P code (MPS) of spindle in multi-spindle	

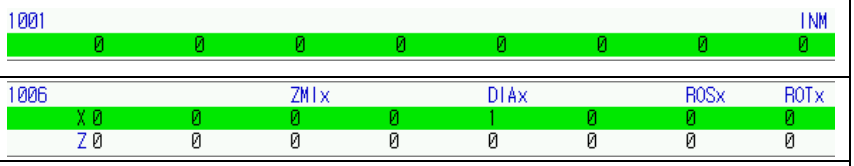
Table 2-18

APPENDIX

Appendix

Appendix 1 Parameters

This chapter mainly introduces CNC state and Value parameters through setting different parameters to realize the different requirements of function. The parameter Value mainly includes the following six types:

<table>
<tr><th colspan="2">Value Types</th><th>Range</th></tr>
<tr><td>(1) Bit</td><td rowspan="3"> 
</td><td rowspan="3">8 digits,0 or 1</td></tr>
<tr><td>(2) Bit axis</td></tr>
<tr><td>(3)Bit spindle type</td></tr>
<tr><td>(4) Word</td><td rowspan="3"> 
</td><td rowspan="3">The setting value range is different according to the variable parameters, refer to the parameter</td></tr>
<tr><td>(5) Word axis</td></tr>
<tr><td>(6) Word spindle type</td></tr>
</table>

For the word axis parameter (3) and (4), the exact Value range is determined by specified parameters.

Each parameter should include the following information:

『Modification authority』: System authority (1st level), Machine authority (2nd level), Equipment management authority (3rd level), Operation authority (4th level), Limited authority (5th level)

『Parameter Type』: Bit, bit axis, word, word axis, Bit axis type, Bit spindle type, Word spindle type

『Validate method』: Become valid immediately or after power-on

『Value Range』: In interval, by enumerating or special judgement)

『Default Setting』: 8 digits in binary system, or 32-digit integral value

Note 1: The 『Data Range』of bit type parameters is 0 or 1.

Note 2: When 『Validate method』 is not stated, the parameter will become valid immediately.

Note 3: When 『Parameter Type』 is not stated, the parameter is of bit type or word type.

Appendix 1.1 Parameter for "Setting"

	#7	#6	#5	#4	#3	#2	#1	#0
0000			**SEQ**			**INI**		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 INI Input unit

0: Metric system

1: Inch system

#5 SEQ whether insert the sequence number automatically

0: No

1: Yes

Note: In EDIT or MDI mode, sequence number can be inserted automatically. The incremental value of sequence number is set in parameter of NO.3216.

Appendix 1.2 Parameters of the Interfaces of Input and Output

0123	**Serial port baud rate (BPS)**

『Modification authority』:Equipment management

『Value Range』: 4800, 9600, 19200, 38400, 57600, 115200

『Defualt Setting』: 115200

	#7	#6	#5	#4	#3	#2	#1	#0
0138		**OWN**						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 OWN When NC Value or the programs are input or output,:

0: whether the covered file information is displayed

1: covered file, is not displayed

	#7	#6	#5	#4	#3	#2	#1	#0
0930						MODBUS	NDSVR	RMEN

『Modification authority』:Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 RMEN Whether use the remote monitoring function

0：YES

1：NO

#1 NDSVR Whether open the Ethernet data communication service

0：Close

1：Open

#2 MODBUS Whether open Modbus communication

0：Close

1：Open

Appendix 1.3 Parameters of Axis Control/Setting Unit

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 INM The least movement increment of linear axis is in:

0: Metric system (metric machine)

1: Inch system (inch machine)

	#7	#6	#5	#4	#3	#2	#1	#0
1002					AZR		DLZ	

『Modification authority』: Machine

『Default Setting』: 0000 0000

#1 DLZ Whether reference setting without dog is valid:

0: Invalid

1: Valid (for all axes)

Note: When DLZ is 0, parameter 1005#1 (DLZx) can set valid/invalid for each axis.

#3 AZR G28 command when the reference point is not set:

0: Reference point return with deceleration dog

1: alarm occurs

Note: The function of reference point return without dog (when parameter 1002#1 (DLZ) is 1 or parameter 1005#1 (DLZx) is 1) is not related to the setting of AZR.

	#7	#6	#5	#4	#3	#2	#1	#0
1004		RPR					ISC	

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#1 ISC Set the least input increment and least command increment

0：0.001mm, 0.001deg or 0.0001inch（IS-B）

1：0.0001mm, 0.0001deg or 0.00001inch（IS-C）

#6 RPR Set the least command increment of the rotation axis and the multiplication of ISC parameter

#7 IPC Set the least command increment of the rotation axis and the multiplication of ISC parameter

00：×1 times

01：×10 times

10：×100 times

	#7	#6	#5	#4	#3	#2	#1	#0
1005					HJZx		DLZx	ZRNx

『Modification authority』: Machine

『Parameter Type』: Bit axis

『Default Setting』: 0000 1000

#0 ZRNx Whether the system alarms if the other traverse commands are specified except G28 before setting the reference point in auto running (AUTO, DNC or MDI).

0: Alarm

1: Not alarm

#1 DLZx Whether setting the reference point free of the link stopper is valid.

0: Invalid

1: Valid

Note: Parameter DLZ (No.1002#1) is valid when it is "0". When DLZ (No.1002#1) is "1", there is no connection with the parameter, and setting the reference point free of the link stopper is valid for all axes.

#3 HJZx After the reference point is set, manually return to the reference point.

0: Use the deceleration link stopper to return to the reference point

1: No connection with the deceleration link stopper, rapidly position in the reference point.

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx		DIAx		ROSx	ROTx

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0, #1 ROTx, ROSx set linear axis or rotary axis

ROSx	ROTx	Content
0	0	Linear axis Metric/inch conversion All coordinate values are of the linear axis type. The stored pitch error compensation is of the linear axis type.
0	1	Rotary axis (type A) No metric/inch conversion Machine tool coordinate value circularly displays based upon the value of the parameter 1260. The relative coordinate value is relevant to the parameters 1008#2, 1008#0, and the absolute coordinate is related with the 1008#0 The stored pitch error compensation is the rotary axis type. Automatically return to the reference point at the direction of the reference point return (G28 and G30), the traverse amount can not exceed one-turn.
1	0	Invalid setting
1	1	Rotary axis (type B) No metric/inch conversion Machine tool coordinate value, relative coordinate value (it is relevant to the parameter 1008#2) and absolute value are linear axes (It can not be circularly display by parameter 1260) The stored pitch error compensation is of the linear axis type.

#3 DIAx sets the traverse amount of each axis

0: specified by the radius

1: specified by the diameter

#5 ZMIx sets the direction of each axis reference point return

0: positive direction

1: negative direction

	#7	#6	#5	#4	#3	#2	#1	#0
1007	RZDx							

『Modification authority』:Machine

『Value Range』：Bit axis

『Default Setting』：0000 0000

#7 RZDx Rotation axis (type A) is in the state of reference point establishment, whether it is the approximate selection direction when reference point returns.

0：Disabled

1：Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
1008						RRLx	RABx	ROAx

『Modification authority』: Machine

『Way of Validating』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 ROAx sets whether the cycle display function of the rotary axis valid.

0: Invalid

1: Valid

Note: ROAx is just valid for the rotary axis and parameter ROTx (No.1006#0) must be 1.

#1 RABx sets the rotation direction of the axis during the absolute command.

0: Rotation direction close to the target

1: Direction specified by the command value coder

Note: RABx is valid only when parameter ROAx is 1.

#2 RRLx Relative coordinate

0: Not cycle as the movement amount of each turn

1: Cycle as the movement amount of each turn

Note 1: RRLx is valid only when ROAx is 1.

1010	Quantity of CNC controlled axes (CCA)

『Modification authority』: Machine

『Validate method』: After power-on

『Value Range』：0～total number

Set the total number of axes which is directly controlled by CNC, the other can be controlled by PLC.

Note: The overall controllable axes numbers are determined by parameter No.8130, and its setting value of this parameter can not be more than the one of the No.8130.

	#7	#6	#5	#4	#3	#2	#1	#0
1015	DWT	WIC						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 WIC The offset measured value of the work piece origin is directly input

0: Only valid for the selected work piece coordinate system

1: Valid for all coordinate systems

#7 DWT When the pause time is specified by P, the Value units are

0: IS-B is 1ms, IS-C is 0.1ms.

1: 1 ms

1020	Programming name of each axis (CAN)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 88（X），89（Y），90（Z），65（A），66（B），67（C），85（U），86（V），87（W）

Set the axial name of each controlled axis.

Note: The same axes names can not be set; U, V and W axes are only enabled in the G code of B set.

1022	The property of each axis in the basic coordinate system

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～7

To ensure the planes of the arc interpolation, the tool offset and the tool nose radius, etc.

G17: X－Y plane

G18: Z－X plane

G19: Y－Z plane

There are four controllable axes: 1 – X basis axis and parallel axis; 2 – Y basis axis and parallel axis; 3 – Z basis axis and parallel axis; 4 – Rotation axis. Only one axis of the basic three axes can be set: X, Y and Z; the parallel axes can be set as two more axes (which is paralleled with the basic axis).

Setting value	Meaning
0	They are neither basic three axes nor the parallel axes,
1	X axis of the basic three axes
2	Y axis of the basic three axes

3	Z axis of the basic three axes
5	Parallel axis of X axis
6	Parallel axis of Y axis
7	Parallel axis of Z axis

1023	Servo axis number of each axis (NSA)

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～99

Set the logic ID number (0～99;0 means that there is no slave station) of the feed servo slave station, its setting value should be corresponding with the one of the servo driver.

Appendix 1.4 Parameter of the Coordinate System

	#7	#6	#5	#4	#3	#2	#1	#0
201	WZR		EWZ	RWO	ZCR	ZCL		

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#2 ZCL After manually return to reference point, the part coordinate system

0: Not cancel

1: Cancel

#3 ZCR After the manual reference point return is completed, the workpiece coordinate system offset value set by G50:

0：Not cancel

1：Cancel

#4 RWO The workpiece coordinate system offset value set by G50 when the coordinate memories after the power is turned on.

0：Clear

1：Restore the memory value from the previous power-off

#5 EWZ The workpiece coordinate system when the power-on coordinate memories.

0：Do not return to G54

1：Return to G54

#7 WZR Work piece coordinate system during resetting

0: Not return to G54

1: Return to G54

	#7	#6	#5	#4	#3	#2	#1	#0
1202					RLC	G50	EWS	EWD

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 EWD The movement direction of the coordinate system caused by the external work piece origin offset amount

0: It is same as the direction specified by the external work piece origin offset amount.

1: It is opposite to the direction specified by the external work piece origin offset amount.

#1 EWS The work piece coordinate system movement amount and the external work piece zero point offset amount

0: Saved in each memorizer

1: Saved in one memorizer (the work piece coordinate system movement amount is same as the external work piece zero point offset amount

#2 G50 When G50 is commanded and the coordinate system is set,

0: Not alarm, but execute G50

1: P/S alarms (No.010), not execute G50

#3 RLC After resetting, the part coordinate system

0: Not cancel

1: Cancel

	#7	#6	#5	#4	#3	#2	#1	#0
1205								MCE

『Modification authority』: Equipment management

『Default Setting』：0000 0000

#0 MCE Whether the coordinate system is memorized with power-on when adapting with the incremental encoder.

0：Do not memory

1：Memory

1206	**The allowable value of the machine coordinate system with the absolute encoder after power on (MER)**

『Modification authority』: Equipment management

『Value Range』： 0～9999

『Parameter Type』：Word axis

『Default Setting』：1000

It is for detecting the offset when the machine coordinate system is set at power on; if it is out of the range, the alarm occurs. The offset isn't detected when it is 0.

1220	**The origin offset amount of each axis external work piece coordinate system (EWO)**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: -9999 9999～9999 9999

This is one parameter to set the origin location of the work piece coordinate system (G54~G59). The parameter is the valid common offset amount for all work piece coordinate system.

Setting unit	IS-B	IS-C	Unit
Linear axis (input in metric system)	0.001	0.0001	mm
Linear axis (input in inch system)	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

1221	**Origin offset amount of each axis in G54 workpiece coordinate system (WO1)**
1222	**Origin offset amount of each axis in G55 workpiece coordinate system (WO2)**
1223	**Origin offset amount of each axis in G56 workpiece coordinate system (WO3)**
1224	**Origin offset amount of each axis in G57 workpiece coordinate system (WO4)**
1225	**Origin offset amount of each axis in G58 workpiece coordinate system (WO5)**
1226	**Origin offset amount of each axis in G59 workpiece coordinate system (WO6)**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: -99 999 999～+99 999 999

This is one parameter to set the origin location of the work piece coordinate system (G54~G59). The parameter is the valid common offset amount for all the work piece coordinate system.

SETTING UNIT	IS-B	IS-C	UNIT
Linear axis (input in metric system)	0.001	0.0001	mm
Linear axis (input in inch system)	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

1240	**Each axis machine coordinate value of the 1st reference point (RF1)**
1241	**Each axis machine coordinate value of the 2nd reference point (RF2)**
1242	**Each axis machine coordinate value of the 3rd reference point (RF3)**
1243	**Each axis machine coordinate value of the 4th reference point (RF4)**

『Modification authority』: Equipment management

『Way of Validating』: 1240 valid after power on; 1241～1243 valid immediately.

『Parameter Type』: Word axis

『Value Range』: -99 999 999～+99 999 999

Set the coordinate values from the 1st to the 4th reference points in the mechanical coordinate system.

SETTING UNITS	IS-B	IS-C	UNITS
Machine in metric system	0.001	0.0001	mm
Machine in inch system	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

1260	**Each turn movement amount of each axis in rotary axis（PRA）**

『Modification authority』: Equipment management

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 1000～9 999 999

Set the movement amount of each turn in rotary axis.

Appendix 1.5 Parameter of the Stroke Detection

Setting unit of stroke parameter Nos.1320~1327 is shown in the following table:

Setting unit	IS-B	IS-C	Unit
Metric machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA	LZR	RL3			LMS		OUT

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 OUT The restricted area of the stroke detection 2 in memory type is set by parameters (No.1322 or No.1323).

0: Internal area

1: External area

#2 LMS Whether the switching signal EXLM of the stroke detection in memory type is valid

0: Invalid

1: Valid

Note: Stroke detection 1 in memory type possesses the parameter of the restricted area set by two groups, signals are switched through the stroke limit in memory type and the set restricted area is selected.

(1) Restricted area I: Parameter No.1320 or No.1321

(2) Restricted area II: Parameter No.1326 or No.1327

#5 RL3 Whether it is valid that the stroke detection 3 releases signal RLS0T3

0: Invalid

1: Valid

#6 LZR After power on before manual reference point return whether detect the stroke 1 in the memory type

0: Detect

1: Not detect

Note: There isn't any connection with the setting when the absolute position encoder is being using, the power is on and the reference point is set. After power on, the stroke is directly detected in memory type.

#7 BFA When the command of overrun memory is sent

0: Alarm after overrun

1: Alarm before overrun

	#7	#6	#5	#4	#3	#2	#1	#0
1301	**PLC**	**OTS**						

『Modification authority』: Equipment management

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#6 OTS Whether output the signal in the overtravel alarm to PLC when the stored stroke detection alarm occurs.

0：Do not output

1：Output

#7 PLC Whether check the stroke before moving

0: No

1: Yes

	#7	#6	#5	#4	#3	#2	#1	#0
1310							**OT3x**	**OT2x**

『Modification authority』: Equipment management

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 OT2X Whether each axis detects the stroke 2 in memory type

0: Not detect

1: Detect

#1 OT3X Whether detect the stroke 3 in memory type in each axis

0: Not detect

1: Detect

1320	**Coordinate value in positive direction boundary of each axis stroke detection 1 in memory type（PC1）**
1321	**Coordinate value in negative direction boundary of each axis**

	stroke detection 1 in memory type（NC1）

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Default Setting』: No.1320 is 99 999 999, No.1321 is －99 999 999

『Value Range』:－99 999 999～99 999 999

Respectively set the coordinate values of boundaries in positive/negative directions in the mechanical coordinate system in stroke detection 1 along each axis in memory type. Set the outside of boundary as the restricted area to tools.

1. The axes specified by diameter are set by diameter value.

2. When (parameter No.1320) < (parameter No.1321) and the limit is infinite, it can not detect the stroke 1 in memory type. (The stroke limit switching signal in memory type is invalid.) If the absolute command is specified, the coordinate value may overflow; the normal movement can not be executed.

1322	Coordinate value in positive direction boundary of each axis stroke detection 2 in memory type（PC2）

1323	Coordinate value in negative direction boundary of each axis stroke detection 2 in memory type（NC2）

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Default Setting』: 0

『Value Range』: －99 999 999～99 999 999

Respectively set the coordinate values of boundaries in positive and negative directions in the mechanical coordinate system in stroke detection 2 along each axis in memory type. The outside or inside of boundary is the restricted area, which is set by parameter OUT (No.1300#0).

Note: The axis specified by diameter must be set by the diameter value.

1324	Coordinate value in positive direction boundary of each axis stroke detection 3 in memory type（PC3）

1325	Coordinate value in negative direction boundary of each axis stroke detection 3 in memory type（NC3）

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Default Setting』:0

『Value Range』: －99 999 999～99 999 999

Respectively set the coordinate values of boundaries in positive and negative directions in the mechanical coordinate system in stroke detection 3 along each axis in memory type. Set inside of the boundary as the restricted area to tools.

Note: The axis specified by the diameter must be set by the diameter value.

1326	**Coordinate value Ⅱ in positive direction boundary of each axis stroke detection 1 in memory type（PC12）**
1327	**Coordinate value Ⅱ in negative direction boundary of each axis stroke detection 1 in memory type（PC12）**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Default Setting』: 0

『Value Range』: －99 999 999～99 999 999

Respectively set the positive and negative boundary coordinate values in stroke detection 1 along each axis in memory type in the machine coordinate system. Set outside of the boundary as the restricted area. When parameter LMS (No.1300#2) is “1”, and the stroke limit switching signal EXLM (G7.6) in memory type is “1”, the restricted area is valid, but it is invalid if it is set by No.1320 and 1321.

1. The axes programmed by the diameter must be set by the diameter value.

2. The parameter is invalid when parameter LMS (No.1320#2) is “0”, or the stroke limit switching signal EXLM (G7.6) in the memory type is “0”. Then, the restricted area set by parameter No.1320 or No. 1321 is valid.

Appendix 1.6 Parameter of the Feedrate

	#7	#6	#5	#4	#3	#2	#1	#0
1401		**RDR**	**TDR**	**RF0**			**LRP**	**RPD**

『Modification authority』: Equipment management

『Default Setting』：0000 0000

#0 RPD Manually rapid run from power on to the reference point return

0: Invalid (JOG speed)

1: Valid

#1 LRP Positioning (G00):

0：Non-linear interpolation positioning

1：Linear interpolation positioning

#4 RF0 When the cutting feedrate override is 0% during rapid traverse

0: tool does not stop moving

1: tool stops moving

#5 TDR During thread cutting or tapping, dry run is:

0: Valid

1: Invalid

#6 RDR To rapid traverse command, dry run is:

0: Invalid

1: valid

	#7	#6	#5	#4	#3	#2	#1	#0
1402						JOV		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 JOV JOG override

0: Valid

1: Invalid (fixed as 100%)

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV		HTG					MIF

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 MIF The minimum unit of F command (the cutting feedrate) of feeding/min

0：1mm/min (input in metric system) or 0.01inch/min (input in inch system)

1：0.001mm/min (input in metric system)or 0.00001inch/min (input in inch system)

#5 HTG The speed command of the spiral interpolation is:

0: Specified by the linear speed of the arc

1: Specified by the linear speed with the linear axis

#7 RTV During thread cutting cycle, the override of the tool run-out is

0: Valid

1: Invalid

#7	#6	#5	#4	#3	#2	#1	#0

1404							F8A	DLF	

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#1 DLF After setting the reference point, manually return to the reference point

0: Move to the reference point (No.1420) at the rapid feedrate

1: Move to the reference point (No.1424) at the manual rapid feedrate

#2 F8A F command range feed/min

0: Set according to parameter MIF (No.1403#0)

1:

SETTING UNITS	UNIT	IS-B	IS-C
Input in metric system	mm/min	1～60000.999	1～24000.999
Input in inch system	inch/min	0.01～2400	0.01～960
Rotary axis	deg/min	1～60000	1～24000

1410	Dry run speed (DRR)

『Modification authority』: Equipment management

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6～15000	6～12000	1000
Machine in inch system	0.1inch/min	6～6000	6～4800	

Set the speed during dry run.

1411	Feedrate in auto mode after power on（IFV）

『Parameter Type』: Word type

『Value Range』: 6～12000

『Default Setting』: 100

SETTING UNITS		VALUE UNITS
Machine in metric system	G98	1 mm/min
	G99	0.001 mm/rev
Machine in inch system	G98	0.1 inch/min
	G99	0.0001 inch/rev

It doesn't require changing the cutting speed in the machine during the processing. And the

cutting feedrate can be set by the parameter, and then the cutting feedrate is not required to be set in the program. But the actual feedrate is limited by parameter NO.1422 which set the maximum cutting feedrate for all axes.

1420 **Each axis rapid movement speed (RTT)**

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	30～100000	6～60000	8000
Machine in inch system	0.1inch/min	30～48000	6～24000	
Rotary axis	1 deg/min	30～100000	6～60000	

Set the rapid movement speed of each axis when the rapid movement override is 100%.

1421 **F0 speed of each axis rapid override（F0R）**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	30～15000	30～12000	400
Machine in inch system	0.1 inch/min	30～12000	30～6000	
Rotary axis	1 deg/min	30～15000	30～12000	

Set the speed when the rapid movement override of each axis is F0.

1422 **Maximum cutting feedrate of all axes (MFR)**

『Modification authority』: Machine

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	

Appendix

Machine in metric system	1mm/min	6～100000	6～60000	8000
Machine in inch system	0.1inch/min	6～48000	6～24000	

Set the maximum cutting feedrate for all axes.

1423 **JOG feedrate of each axis (JFR)**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6～60000		1000
Machine in inch system	0.1inch/min			
Rotary axis	1 deg/min			

Set the feedrate of each axis during continually manual feeding (JOG feeding), the actual feedrate is limited by parameter NO.1422 (the maximum cutting feedrate of all axes).

1424 **Manual rapid speed of each axis (MRR)**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNIT	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Metric machine	1 mm/min	30～100000	30～60000	8000
Inch machine	0.1 inch/min	30～48000	30～24000	
Rotary axis	1 deg/min	30～100000	30～60000	

Set the speed of each axis manual rapid movement when rapid movement override is 100%.

Set the maximum speed of MPG feeding.

Note: If it is set as 0, use the setting value of parameter 1420.

1425 **FL speed of each axis reference point return (FLR)**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	6～15000	6～12000	200
Machine in inch system	0.1 inch/min	6～12000	6～6000	
Rotary axis	1 deg/min	6～15000	6～12000	

After deceleration is performed, set the speed (FL speed) of each axis during the reference point return.

1428 **Reference point return speed along each axis (RPF)**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE	DEFAULT SETTING
Machine in metric system	1 mm/min	0，6～60000	5000
Machine in inch system	0.1 inch/min		
Rotary axis	1 deg/min		

Set the situation of the reference point return used the deceleration block, alternatively, the rapid traverse rate based upon the reference point return regardless of the state of reference point. When the parameter value sets to 0, parameter №1421 is enabled.

1434 **The Max. feedrate of the Manual MPG along each axis (HMF)**

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE	DEFAULT SETTING
Machine in metric system	1 mm/min	0，6～60000	5000
Machine in inch system	0.1 inch/min		
Rotary axis	1 deg/min		

Set the Max. feedrate of the manual MPG of each axis. When its setting is 0, the setting value of parameter №1424 is enabled.

1466	The retracting feedrate during the thread cutting (FRT)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS Machine in metric system	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in inch system	1 mm/min	6～100000	6～60000	8000
SETTING UNITS	0.1 inch/min	6～48000	6～24000	

Set the feedrate of end-retraction operation of the thread cutting machining. When this parameter sets to “0”, that is, the speed of long axis is performed the end-retraction operation.

Appendix 1.7 Parameter of Control of Acceleration and Deceleration

	#7	#6	#5	#4	#3	#2	#1	#0
1601				RTO				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 RTO During rapid running, the block is

0: No overlapping

1: Overlapping

	#7	#6	#5	#4	#3	#2	#1	#0
1610			THLX	JGLx				

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Default Setting』：0000 0000

#4 JGLx The acceleration/deceleration for the manual feed

0：Exponential acceleration/deceleration

1：Linear acceleration/deceleration after interpolation

#5 THLX The acceleration/deceleration of the end-retraction operation in the thread cutting machining:

0：Exponential acceleration/deceleration

1：Linear acceleration/deceleration

1620	Time constant T of linear acceleration and deceleration of each axis rapid movement (TT1)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: 0～4000 ms

『Default Setting』: 100

Set the time constant of acceleration and deceleration during rapid movement.

1622	Time constant of acceleration and deceleration during cutting and feeding after each axis interpolation (ATC)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: 0～4000 ms

『Default Setting』: 100

Set the acceleration and deceleration of each axis cutting and feeding in index type, or the time constant of acceleration and deceleration in linear type after interpolation.

And the detailed type is set by parameter CTLx (NO.1610#0). If CTLx sets the acceleration and deceleration in linear type after linear interpolation, the maximum time constant of acceleration and deceleration is limited in 512ms and even it exceeds 512ms, it is still dealt as 512ms.

Note: Except the special usage of the parameter, all axes must be set as the same time constant. If the different time constants are set, the correct linear or circular can't be shaped.

1624	Time constant of acceleration and deceleration of each axis JOG feeding after interpolation (JET)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 0～4000ms

『Default Setting』: 100

Set the acceleration and deceleration in index type of each axis JOG feeding, and the time constant of acceleration and deceleration in linear type after interpolation.

The detailed type is set by parameter JGLx (NO.1610#4). If JGLx sets the acceleration and deceleration in linear type after interpolation, the maximum time constant of acceleration and deceleration is limited in 512ms and even it exceeds 512ms, it is dealt as 512ms.

1625	FL speed of acceleration and deceleration in index type during each axis JOG feeding (FLJ)

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Metric machine	1 mm/min	0，6～15000	0，6～12000	30
Inch machine	0.1 inch/min	0，6～12000	0，6～6000	30
Rotary axis	1 deg/min	0，6～15000	0，6～12000	30

Set the low limit speed (FL speed) of acceleration and deceleration in index type during each axis JOG feeding.

1626	Time constant of acceleration and deceleration during each axis thread cutting cycle (TET)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: 0～4000ms

『Default Setting』: 100

Set the time constant of acceleration and deceleration in linear and index types during each axis thread cutting cycle.

1627	FL speed of acceleration and deceleration in index type during each axis thread cutting cycle (FLT)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	0，6～15000	0，6～12000	30
Machine in inch system	0.1 inch/min	0，6～12000	0，6～6000	30

Set low limit speed (FL speed) of acceleration and deceleration in index type during each axis thread cutting cycle.

1628	The acceleration/deceleration time constant of the end-retraction motion in the thread cutting cycle along each axis (TST)

『Modification authority』: Equipment management

『Parameter Type』: Word axis

『Value Range』: 0～4000ms

『Default Setting』: 0

Set the acceleration/deceleration time constant of end-retraction short axis when the thread cutting cycle of each axis is performed; when this parameter setting value is “0”, use the No.1626 parameter value (0～4000ms).

Appendix 1.8 Parameter of Servo and Backlash Compensation

	#7	#6	#5	#4	#3	#2	#1	#0
1800	BDEC	BD8						

『Modification authority』: Machine

『Default Setting』: 1000 0000

#6 BD8: Impulse output frequency of the backlash compensation

0: Compensate at the frequency set by parameter #1853

1: Compensate at 1/8 of frequency set by parameter #1853

#7 BDEC:Backlash compensation mode

0: Fixed pulse frequency output, which is set by parameters #1853 and #1800.6.

1: Pulse frequency output based on the acceleration and deceleration characteristics.

	#7	#6	#5	#4	#3	#2	#1	#0
1811						POD		

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#2 POD Selecting output directions of each axis pulse

0: Not inversed

1: Inversed

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx				

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#4 APZx The mechanical position and the absolute position detector position during using the absolute position detector

0: Not consistent

1: Consistent

Note: When use the absolute position detector, during the initial setting or after changing the absolute position encoder, the parameter must be set as 0, and connect power supply, again after power off and manually return to the reference point. Therefore, the mechanical position consists with that of the position encoder, and the parameter will be auto set as 1.

#5 APCx Position encoder

0: Not use the absolute position detector

1: Use the absolute position detector (the absolute pulse encoder)

1816	Each axis detection multiply ratio (DMR)

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 1~32767

『Default Setting』: 2

The detection multiply ratio (DMR) of each axis is set

1820	Command multiply ratio of each axis（CMR）

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 1~32767

『Default Setting』: 2

Gear ratio output by each axis＝CMR/ DMR

Detection unit=minimum movement unit/ CMR

The relations between the setting units and the minimum movement units:

		IS－B		IS－C	
	Input	Least input increment	Least command increment	Least input increment	Least command increment
Metric		0.001mm（Diameter）	0.0005mm	0.0001mm（Diameter）	0.00005mm

		0.001mm（Radius）	0.001mm	0.0001mm（Radius）	0.0001mm
	Inch	0.0001 inch（Diameter）	0.0005mm	0.00001 inch（Diameter）	0.00005mm
		0.0001 inch（Radius）	0.001mm	0.00001 inch（Radius）	0.0001mm
Inch machine	Metric	0.001mm（Diameter）	0.00005 inch	0.0001mm（Diameter）	0.000005 inch
		0.001mm（Radius）	0.0001 inch	0.0001mm（Radius）	0.00001 inch
	Inch	0.0001 inch（Diameter）	0.00005 inch	0.00001 inch（Diameter）	0.000005 inch
		0.0001 inch（Radius）	0.0001 inch	0.00001 inch（Radius）	0.00001 inch
Rotary axis		0.001deg	0.001deg	0.0001deg	0.0001deg

1851 **Backlash compensation value of each axis (BCV)**

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: -9999～+9999 (Detection unit)

『Default Setting』: 0

Set the backlash compensation value of each axis.

After connecting power supply, it compensates the backlash at the first time when the machine moves in the direction opposite with that of the reference point return.

Detection units are related with parameter No.1820 (command multiply ratio CMR) and the minimum movement units, about the relations between the setting units and the minimum movement units, refer to parameter No.1820 introduction.

1853 **The setting value of reverse interval compensation pulse frequency**

『Modification authority』: Machine

『Parameter Type』: Word

『Value Range』: 1~32

『Default Setting』: 12

The setting value of reverse interval compensation pulse frequency (1~32)

2071 **Each axis backlash acceleration and deceleration valid time constant (BAT)**

『Modification authority』: Machine

『Parameter Type』: Word axis

『Value Range』: 0～100 ms

『Default Setting』: 40

Set each axis backlash acceleration and deceleration valid time constant.

Appendix 1.9 Parameter of Input/Output

	#7	#6	#5	#4	#3	#2	#1	#0
3001						RWM		

『Modification authority』: Machine

『Default Setting』: 0000 0000

#2 RWM Whether output the rewinding signal in the program back within the program memory (RWD)

0：Do not output

1：Output

	#7	#6	#5	#4	#3	#2	#1	#0
3003	ESP					ITX		ITL

『Modification authority』: Machine

『Default Setting』: 1000 0000

#0 ITL To interlock the signal of the overall axes

0：Disabled

1：Enabled

#2 ITX To interlock the signal of each axis

0：Disabled

1：Enabled

#7 ESP External emergency stop alarm input signal (X0.5)

0: When the signal is 0 (low level), emergency stop alarms

1: When the signal is 1 (high level), emergency stop alarms

	#7	#6	#5	#4	#3	#2	#1	#0
3004			OTH					BSL

『Modification authority』: Machine

『Default Setting』: 0010 0000

#0 BSL Block starts the interlocking signal and the cutting block starts the interlocking signal

0：Disabled

1：Enabled

#5 OTH Overtravel limit signal

0: Check

1: Not check

Note: After the overtravel alarm occurs, this parameter is altered to 1 (without detection), and the alarm will not be cleared pressing the resetting again; it is necessary to move inside the stroke by hand; and then set this parameter to 0, the alarm is eliminated accordingly.

	#7	#6	#5	#4	#3	#2	#1	#0
3006						EPS	EPN	GDC

『Modification authority』: Machine

『Default Setting』: 0000 0000

#0 GDC Deceleration signal of the reference point return

0: Use X signal

1: Use G196 (X signal is invalid)

#1 EPN In the external workpiece number index, select the signal for specifying the workpiece.

0：Usable signals PN1～PN16

1：Usable extension signals EPN0～EPN13

#2 EPS The start signal at the external workpiece number index

0：Use the automatic operation start signal ST

1：Use the external workpiece index start signal EPNS

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

『Modification authority』: Machine

『Default Setting』: 0000 0000

#2 XSG The X address is distributed to the skip signal and measurement position arrival signal

0：It is the fixed address

1：Changeable any X addresses

	#7	#6	#5	#4	#3	#2	#1	#0
3009			DECx					

『Modification authority』: Machine

『Parameter Type』: Bit axis

『Default Setting』: 0010 0000

#5 DECx: Deceleration signal of the reference point return

0: When the signal is 0 (low level), decelerate.

1: When the signal is 1 (high level), decelerate.

3010	Dwell time of the gating signals MT, TF and SF (MFT)

『Modification authority』: Machine

『Value Range』: 16 ms～32767 ms

『Default Setting』: 16

Set the time from sending codes M, S, T and B, till MF, SF, TF and BF being sent.

3011	Minimum width (MAW)of completion signals (FIN)of M, T and S (MAW)

『Modification authority』: Machine

『Value Range』: 16 ms～32767 ms

『Default Setting』: 16

Set the minimum width of the completion signals (FIN) of M, S, T and B function.

Note: The time is set by 8ms, if its setting value does not the multiplication of the 8; the carry-bit is multiplication of the 8.

3012	Address to be assigned to skip signals

『Modification authority』: Machine

『Value Range』: 0～127

『Default Setting』: 0

Set the skip signal to assort the X address and measure the address of the position arrival signal (0～127).

3013	X Address to be assigned to reference position return deceleration signals

『Modification authority』: Machine

『Value Range』: 0～127

『Default Setting』: 3

Set the X address to be assigned to the reference position return deceleration signal for each axis （0～127）。

3014	Bit position to be assigned to reference position return

	deceleration signals

『Modification authority』: Machine

『Value Range』: 0～7

『Default Setting』: 0

Set the X bit position to be assigned to the reference position return deceleration signal（*DECn）for each axis（0～7）。

3017	Output time of the resetting signal (RST)

『Modification authority』: Machine

『Value Range』: 0～255

『Default Setting』: 32

Set the dwell time when the resetting signal RST is output.

RST signal output time =resetting time + the parameter value X 16ms.

3019	Distribute the address of tool compensation value write-in signal

『Modification authority』: Machine

『Value Range』: 0～127

『Default Setting』: 0

Set the address of tool compensation value write-in signal for distributing the X address.

3020	Distribute the bit address of the skip signal X address

『Modification authority』: Machine

『Value Range』: 0～7

『Default Setting』: 0

Set the bit address for distributing the skip signal X address.

3021	Distribute the bit address of the multistep skips signal SKIP2

『Modification authority』: Machine

『Value Range』: 0～7

『Default Setting』: 0

Set the bit address for distributing the multistep skips signal SKIP2

3022	Distribute the bit address of the multistep skips signal SKIP3

『Modification authority』: Machine

『Value Range』: 0～7

『Default Setting』： 0

Set the bit address for distributing the multistep skips signal SKIP3

3023	Distribute the bit address of the multistep skips signal SKIP4

『Modification authority』: Machine

『Value Range』： 0～7

『Default Setting』： 0

Set the bit address for distributing the multistep skips signal SKIP4

3030	Allowable digits of M code（MCB）

『Modification authority』: Machine

『Value Range』: 2～8

『Default Setting』: 4

Set the allowable digits of M code.

3031	Allowable digits of S code（SCB）

『Modification authority』: Machine

『Value Range』: 1～5

『Default Setting』: 4

Set the allowable digits of S code.(Maximum 5 digits in S code is allowed).

3032	Allowable digits of T code（TCB）

『Modification authority』: Machine

『Value Range』: 2～8

『Default Setting』: 4

Set the allowable digits of T code.

3033	Allowable number of digits for the B code (BCN)

『Modification authority』: Machine

『Value Range』： 0～8

『Default Setting』： 0

The allowable bit number (0～8) of B code (The 2nd miscellaneous function)

3050	I/O unit quantity (IOMAX) of the system control

『Modification authority』: Machine

『Value Range』: 0～4

『Default Setting』: 0

Set the I/O unit quantity (up to 4) controlled by system.

3051	The logic ID number (IOID1) of system control I/O unit 1

『Modification authority』: Machine

『Value Range』: 0,100~110

『Default Setting』: 0

Set the logic ID number (0 means that this I/O unit disconnects with the GSKLink) of the system control I/O unit 1.

3052	The logic ID number (IOID2) of system control I/O unit 2

『Modification authority』: Machine

『Value Range』: 0,100~110

『Default Setting』: 0

Set the logic ID number (0 means that this I/O unit disconnects with the GSKLink) of the system control I/O unit 2.

3053	The logic ID number (IOID3) of system control I/O unit 3

『Modification authority』: Machine

『Value Range』: 0,100~110

『Default Setting』: 0

Set the logic ID number (0 means that this I/O unit disconnects with the GSKLink) of the system control I/O unit 3.

3054	The logic ID number (IOID4) of system control I/O unit 4

『Modification authority』: Machine

『Value Range』: 0,100~110

『Default Setting』: 0

Set the logic ID number (0 means that this I/O unit disconnects with the GSKLink) of the system control I/O unit 4.

3060	The logic ID number (GWID) of the system gateway control

『Modification authority』: Machine

『Value Range』: 0,200~254

『Default Setting』： 0

This parameter setting system controls the logic ID number of the gateway. (0 means not use the gateway

	#7	#6	#5	#4	#3	#2	#1	#0
3061							GWP	GWC

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』： Bit

『Default Setting』： 0000 0000

#0 GWC Whether the gateway data uses the CRC verification

0：Disabled

1：Enabled

#1 GWP Whether the gate data uses the communication agreement

0：Disabled

1：Enabled

Appendix 1.10 Parameter of Display and Editing

	#7	#6	#5	#4	#3	#2	#1	#0
3101				BGD				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 BGD Background editing selects the programs selected at the foreground

0: Editable

1: Unedited

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC	DAL	DRC	DRL				MCN

『Modification authority』:Machine

『Default Setting』: 1100 0000

#0 MCN Display the machine position

0: Display based on the output units

(There isn't any connection with the metric system or the inch system, the metric machine displays as the metric units, the inch machine displays as the inch units.)

1: Display based on the input units

(When it is input in the metric system, display in the metric system; when it is input in the inch system, display in the inch system)

#4 DRL Display the relative position

0: Display the actual position including the tool offset (T serial)

1: Display the programming position without the tool offset (T serial)

Note: In T serial, the movement coordinate system compensates the tool appearance, (parameter LGT (NO.5002#4) is 0), display the programming position which ignores the tool compensation (the parameter is set as 1). However, the programming position without the tool appearance compensation value can not display.

#5 DRC Display the relative position

0: Display the actual position including the tool nose radius compensation (T serial)

1: Display the programming position without the tool nose radius compensation (T serial)

#6 DAL Display the absolute position

0: Display the actual position including the tool offset (T serial)

1: Display the programming position without the tool offset (T serial)

Note: In T serial, the movement coordinate system compensates the tool appearance (parameter LGT (NO.5002#4) is 0), and display the programming position which ignores the tool compensation (the parameter is set as 1). However, the programming position without the tool appearance compensation value can not display.

#7 DAC Display the absolutely position

0: Display the actual position including the tool nose radius compensation (T serial)

1: Display the programming position without the tool nose radius compensation (T serial)

	#7	#6	#5	#4	#3	#2	#1	#0
3107					REV	DNC		

『Modification authority』: Equipment management

『Default Setting』: 0001 0000

#2 DNC Whether clear display of DNC running programs during resetting

0: Not clear

1: Clear

#3 REV Display the actual speed in feeding/rev mode

0: mm/min or inch/min

1: mm/rev or inch/rev

	#7	#6	#5	#4	#3	#2	#1	#0
3110						AHC		

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0100

#2 AHC Whether the alarm resume can be cleared by soft keys

0: Yes

1: No

	#7	#6	#5	#4	#3	#2	#1	#0
3111	NPA							

『Modification authority』: Equipment management

『Default Setting』: 1000 0000

#7 AHC Whether switch to alarm/information window when alarm occurs or information is input:

0: No

1: Yes

	#7	#6	#5	#4	#3	#2	#1	#0
3114								IPC

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 IPC On the current interface, press the function keys

0: Switch into the interface

1: Not switch into the interface

	#7	#6	#5	#4	#3	#2	#1	#0
3115								NDPx

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 NDPx Whether displays the current position

0：YES

1：NO

	#7	#6	#5	#4	#3	#2	#1	#0
3200		PSR		NE9				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 NE9 Whether forbid the operations, such as program editing, deletion, modification and copy, etc. followed with the program number 9000.

0：Allow

1：Forbid

#6 PSR Whether allow loading and checking the protected program

0：Forbid

1：Allow

	#7	#6	#5	#4	#3	#2	#1	#0
3202			CPD					NE8

『Modification authority』: Equipment management

『Default Setting』: 0010 0000

#0 NE8 Whether forbid the operations, such as program editing, deletion, modification and copy, etc. of the program number 8000～8999.

0：Allow

1：Forbid

#5 CPD When NC program is deleted, confirm information and keys

0: Not display

1: Display

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL	MER						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 MER When the single block runs in MDI mode, after the last block is executed in the program, whether the executed programs are

0: Not deleted

1: Deleted

Note: Even MER is 0, when "%" (end code) is read in and executed, the program is also deleted ("%"is auto inserted at the end of the program).

#7 MCL Whether delete the programs edited in MDI mode through resetting

0: Not delete

1: Delete

	#7	#6	#5	#4	#3	#2	#1	#0
3209								MPD

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 MPD When the subprogram is executed, whether display the main program number

0: Not display

1: Display

3212	**NE9 needs the program quantity (CN9) protected from program**

『Modification authority』: Equipment management

『Value Range』: 0～999

『Default Setting』： 0

The programs after the No.9000 to be protected are set on the quantity, the program number protection range is 9000~9000+(No.3212), 0 is the overall protections (0～999).

3216	Increment value (INC) during the serial number being auto inserted (INC)

『Modification authority』:Equipment management

『Value Range』: 1～9999

『Default Setting』: 10

When the serial number (parameter SEQ (NO.0000#5) is 1) is auto inserted, it is the increment value of the serial number in each block.

3281	Language displayed on the screen (LANG)

『Modification authority』： Machine

『Value Range』： 0～1

『Default Setting』： 1

0：English 1：Chinese

3282	Reminding days before power off in the limited time (NDAYS)

『Modification authority』： Machine

『Value Range』： 1～30

『Default Setting』： 3

Appendix 1.11 Parameter of Programming

	#7	#6	#5	#4	#3	#2	#1	#0
3401		GSB				NCK		DPI

『Modification authority』: Equipment management

『Default Setting』: 0000 0001

#0 DPI The address is with the decimal point, but when the decimal point is omitted, the setting is as below:

0: Take them as the minimum setting units

1: Take them as the units of mm, inch and sec

#2 NCK During grammar checking, there are same N numbers

0: Alarm

1: Not alarm

#6 GSB Set the G code format

0：G code system A

1：G code system B

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR		FPM	G91			G01

『Modification authority』: Equipment management

『Default Setting』: 0101 0000

#0 G01 Mode during connecting the power supply

0: G00 mode (orientation)

1: G01 mode (linear interpolation)

#3 G91 In the G code system B, the system defaults as:

0：G90 mode (Absolute command)

1：G91 mode (Incremental command)

#4 FPM System defaults after power on

0: Feeding/rev

1: Feeding/min

#6 CLR Press the resetting key on MDI panel, the external resetting signal and the emergency stops, G code mode and the feedrate are

0: Hold mode

1: Switched to the power on state

#7 G23 when the power supply is connected, it is

0: G22 mode (Check the memory stroke)

1: G23 mode (Not check the memory stroke)

	#7	#6	#5	#4	#3	#2	#1	#0
3403		AD2	CIR	RER				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 RER During arc interpolation, when R goes over the minor finishing point and isn't in the arc, and the radius doesn't exceed error:

0: Calculate the new radius, the path is semicircle

1: P/S alarms

#5 CIR In arc interpolation commands (G02, G03), there are no distance (I, J and K) from the starting point of the command to the center, and the arc radius isn't commanded,

either.

0: Linear interpolation moves to the finishing point

1: P/S alarms

#6 AD2 In one block, two or two more same addresses are commanded

0: The following commands are valid.

1: The program is taken as wrong, P/S alarms.

Note:	**It alarms when the parameter is 1and two or two more G codes of one group are commanded in one block.**

	#7	#6	#5	#4	#3	#2	#1	#0
3404	**M3B**	**EOR**	**M02**	**M30**				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 M30 During auto running, process M30 command

0: return to the beginning of the program.

1: doesn't return to the beginning of the program.

#5 M02 During auto running, process M02 command

0: return to the beginning of the program.

1: doesn't return to the beginning of the program.

#6 EOR During executing the program, read in "%" (program end)

0: P/S alarms (stop auto running, display alarm state)

1: Not alarm (auto running stops, the system resets)

Note:	**When performing the "%" (end-of-program), CNC resets instead of closing the miscellaneous function output.**

#7 M3B The quantity of M codes which can be commanded in one block

0: One

1: Maximum three

	#7	#6	#5	#4	#3	#2	#1	#0
3405			**DDP**					**AUX**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 AUS In the 2nd miscellaneous function, the command counter decimal point input or the command with decimal point, as well the override corresponding to the command value output

0：The metric input is identical with the inch input

1：The override set by inch input sets as the 10 times of the override for the metric input

#5 DDP The angle command is directly input based upon the drawing dimension

0：Common specification

1：Command supplementary angle

3410	Circular radius allowable error (CRE)

『Modification authority』: Equipment management

『Value Range』: 0～9999 9999

Setting unit	IS－B	IS－C	Unit
Input in mm	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

『Default Setting』:0

Set the allowable error value of arc interpolation (G02, G03) starting point radius and its finishing point radius. P/S alarms when arc interpolation radius error is more than the limit value.

Note: When the setting value is 0, it doesn't require checking the arc radius error.

3411	M code 1 for stopping the buffer (BLKM1)
3412	M code 2 for stopping the buffer (BLKM2)
3413	M code 3 for stopping the buffer (BLKM3)
3414	M code 4 for stopping the buffer (BLKM4)
3415	M code 5 for stopping the buffer (BLKM5)
3416	M code 6 for stopping the buffer (BLKM6)
3417	M code 7 for stopping the buffer (BLKM7)
3418	M code 8 for stopping the buffer (BLKM8)

『Modification authority』: Equipment management

『Validate method』: Immediately

『Parameter Type』: Word

『Value Range』: 0～9999

『Default Setting』: 0

This parameter sets the M code for stopping the buffer. Before ending the treatment of the M function at the side of the machinery, it is necessary to perform the operation treatment specified by M code by machinery, and then set this code.

	#7	#6	#5	#4	#3	#2	#1	#0
3450								AUP

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 AUP In the 2nd miscellaneous function command, the counter decimal point input, the command with decimal point and the negative value command

0：Disabled

1：Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
3453								CRD

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 CRD Chamfering/corner R is valid (the parameter CCR(No.8134)="1"）

0: Chamfering/corner R is enabled.

1: Direct drawing dimension programming is enabled.

3460	Address for the second miscellaneous function (BCA)

『Modification authority』: Equipment management

『Validate method』: Immediately

『Parameter Type』: Word

『Value Range』: 0,65~67，85~87

『Default Setting』: 0

The address(0,65~67，85~87) for the second miscellaneous function, when it is 0, the second miscellaneous function is off.

Appendix 1.12 Parameter of Screw Pitch Error Compensation

	#7	#6	#5	#4	#3	#2	#1	#0
3605								BDPx

『Modification authority』: Machine

『Validate method』: After power-on

『Value Range』: Bit axis

『Default Setting』: 0000 0000

#0 BDPx Whether use the bi-directional pitch error compensation

0：NO

1：YES

3620	Screw pitch error compensation number in each axis reference point (NPR)

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～1023

『Default Setting』: 0

3621	Number of the furthest screw pitch error compensation point of each axis in negative direction (NEN)

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～1023

『Default Setting』: 0

The parameter sets the number of the furthest screw pitch error compensation point of each axis in negative direction.

3622	Number of the furthest screw pitch error compensation point of each axis in positive direction (NEP)

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～1023

『Default Setting』: 0

The parameter sets the number of the furthest screw pitch error compensation point of each axis in positive direction.

Note: The parameter setting value should be greater than that of parameter NO.3620.

3623	Each axis screw pitch error compensation override (PCM)

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～100

『Default Setting』: 0

Set the override of screw pitch error compensation along each axis.

If the override is set as 1, the detection unit is same as that of compensation.

If the override is set as 0, the override is same as one when it is set as 1.

3624 **Each axis screw pitch error compensation point interval (PCI)**

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Default Setting』: 0～9 999 999

『Default Setting』: 0

Setting unit	IS－B	IS－C	Unit
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

The screw pitch compensation points are distributed in equal interval, and the interval value of each axis is set respectively. The minimum value of the interval is limited and set by the following formula: the minimum value = the maximum feedrate (rapid feedrate) / 7500.

Unit: Screw pitch compensation minimum interval: mm, inch and deg.

Maximum feedrate: mm/min, inch/min and deg/min.

For example: When the maximum feedrate is 15000mm/min, the minimum value of the screw pitch error compensation interval is 2mm.

But, according to the setting override, when the absolute value of the compensation point value exceeds 100, the interval of the compensation point is magnified by the override which is calculated by the following formula.

Override = Max compensation amount (absolute value)/128 (round up the digits after the decimal point)

Screw pitch compensation minimum interval = Value, which is obtained from the above maximum feedrate X override.

Note: The unit of the screw pitch compensation value is same as that of the detection.

The detection unit is relative with parameter No.1820 (command magnify ratio CMR) and the minimum movement unit, about the relation between the setting units and the minimum movement units, refer to the introduction of parameter No.1820.

3626	The compensation point (NPN) of the closest negative side for the bi-directional pitch error compensation

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Default Setting』: 0～1023

『Default Setting』: 0

When using the bi-directional pitch error compensation, set the closest negative side compensation point number when the tool moves along with the negative direction.

3627	The pitch error compensation value (PCD) in the reference point moves to the reference point from the negative direction of the origin direction return

『Modification authority』:Machine

『Validate method』: After power-on

『Parameter Type』: Word axis

『Default Setting』: -32768～32767

『Default Setting』: 0

When the origin direction is set as positive/negative direction; the pitch error compensation value in the reference point when the movement is set from negative/positive direction based upon absolute value.

3628	The setting value of the pitch compensation pulse frequency (NPF)

『Modification authority』:Machine

『Parameter Type』: Word

『Default Setting』:1～32

『Default Setting』: 8

The setting value of the pitch compensation pulse frequency

Appendix 1.13 Parameter of the Spindle Control

	#7	#6	#5	#4	#3	#2	#1	#0
3700						CSB		CSC

『Modification authority』: Equipment management

『Parameter Type』: Bit type

『Default Setting』: 0000 0000

#0 CSC Whether the coordinate value is cleared (Bit 2 of parameter 3700 sets to 0, this parameter is enabled) when the CS outline control shifts to spindle mode.

0：Keep

1：Clear

#2 CSB Whether the coordinate system is automatically set up when CS outline control shifts to the position mode

0：Disabled

1：Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
3703					**MPP**			

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#3 MPP Whether replaces the signal SWS to perform the spindle selection by program command in the multi-axis control.

0：NO

1：YES

	#7	#6	#5	#4	#3	#2	#1	#0
3704	**SCS3**	**SCS2**						

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#6 SCS2 Whether Cs contour control of the 2nd spindle is

0: Invalid

1: Valid

#7 SCS3 Whether Cs contour control of the 3rd spindle is

0: Invalid

1: Valid

Note: Parameters SCS2 and SCS3 can be enabled by using the Cs outline control (that is, bit 2 of parameter No.8133 (SCS) is "1")

	#7	#6	#5	#4	#3	#2	#1	#0
3705				**EVS**				

『Modification authority』: Equipment management

『Default Setting』： 0000 0000

#4 EVS For S command, use spindle control function (spindle analog output or spindle serial output)

0: Not output S code and SF

1: Output S code and SF

	#7	#6	#5	#4	#3	#2	#1	#0
3706						MPA		

『Modification authority』: Equipment management

『Default Setting』： 0000 0000

#2 MPA In the multi-spindle control, when the spindle selection of the address P is set, and when the P does not specify with the S command:

0：Alarm issues（PS5303）

1：Use the last P specified by S_ P_;. After the power is turned on, use the value of parameter (№3775) when never ever specifies the P.

	#7	#6	#5	#4	#3	#2	#1	#0
3708		TSO				SSC	SAT	SAR

『Modification authority』: Equipment management

『Default Setting』: 0000 0011

#0 SAR Whether check the spindle speed reaching signal

0: Not check

1: Check

#1 SAT Whether check the spindle speed reaching signal when the thread cutting block is begun to be executed.

0: Check or not, which is set by parameter SAR (NO.3708#0)

1: Must check, which isn't connected with parameter SAR

Note: When the thread cutting block is continuously executed, the spindle speed reaching signal isn't checked in the thread cutting block after the 2nd block.

#2 SSC Whether check the spindle speed when performs the cutting feed

0：Do not check

1：Check

#6 TSO Whether the spindle override is valid during thread processing or tapping cycle

0: Invalid (fixed as 100%)

1: Valid

Note: In rigid tapping, the override is fixed as 100%, and there isn't any connection with the

setting of the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709						MSI		SAM

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 SAM Times of sampling in spindle average speed

0: Four times (Generally it is set as 0)

1: One time

#2 MSI SIND signal is valid during multi-spindle control

0: It is only valid for the 1st spindle. (SIND signal of the 2nd spindle becomes invalid.)

1: No matter whether each spindle is selected or not, it is valid for all spindles. (Each spindle has its own SIND signal.)

3710	Spindle number control of CNC (CCS)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 1～3

『Default Setting』: 1

Set the spindle number of the CNC control

	#7	#6	#5	#4	#3	#2	#1	#0
3713		MPC						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 MPC In the multi-spindle, when the spindle selection is performed based upon the program command of address P, whether is automatically perform position encoder feedback shifting used in the thread cutting or feed/rev. based upon the selected spindle

0：Do not shift

1：Shift

3717	The amplifier number of each spindle (NSS)

『Modification authority』: System

『Validate method』: After power-on

『Parameter Type』: Word axis

『Value Range』： -4～99

『Default Setting』： 1

Set the amplifier number distributing to each spindle

Set value by parameter	Corresponding interface	Remark
0	Disconnect the spindle amplifier interface	
1~99	Spindle connects the logic ID number by GSKLink	The setting value is identical with the servo spindle logic ID number
-1～-4	Four groups analog value output ports of the spindle interfaces 1 and 2 on the corresponding the I/O unit 1	It is used in the frequency-conversion spindle
-11～-14	Four groups analog value output ports of the spindle interfaces 1 and 2 on the corresponding the I/O unit 2	
-21～-24	Four groups analog value output ports of the spindle interfaces 1 and 2 on the corresponding the I/O unit 3	
-31～-34	Four groups analog value output ports of the spindle interfaces 1 and 2 on the corresponding the I/O unit 4	

3720 **Revolution of each spindle coder（CNT）**

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』： Word axis

『Value Range』： 100～99999999

『Default Setting』： 1024

The revolution of each spindle coder is set

3721 **Number of position coder gear teeth for each spindle (GOE)**

『Modification authority』: Machine

『Parameter Type』： Word axis

『Value Range』： 1～9999

『Default Setting』： 1

Set the number of position coder gear teeth for each spindle during the speed control (feeding per revolution, thread cutting, etc).

3722	Number of gear teeth for each spindle (GOS)

『Modification authority』: Machine

『Parameter Type』： Word axis

『Value Range』： 1～9999

『Default Setting』： 1

Set the number of gear teeth for each spindle during the speed control (feeding per revolution, thread cutting, etc).

3723	Channel number corresponding to each spindle coder (CSE)

『Modification authority』: Machine

『Validate method』: After power-on

『Parameter Type』： Word axis

『Value Range』： 0～2

『Default Setting』： 0

Set the channel number corresponding to each spindle coder。

Value set by the parameter	Corresponding channel interface	
0	The data of spindle encoder is transmitted from GSKLink	It is used by using the GSKLink spindle and without external encoder.
1	With the 1st coder channel interface	It is used by using the external encoder.
2	With the 2nd coder channel interface	

3730	Increment adjustment Value of the spindle speed analog output (AGS)

『Modification authority』:Machine

『Parameter Type』: Word spindle

『Default Setting』: 1000

『Value Range』: 500～2000

『Value unit』: 0.1%

Set the increment adjustment Value of the spindle speed analog output. (Adjusting method)

（1） Set the standard setting value 1000,

（2） Command the spindle speed when the spindle speed analog output maximum voltage is 10V.

（3） Measure the output voltage.

（4） Set the value in the following formula in parameter No.3730:

$$\text{setting value} = \frac{10(\text{V})}{\text{measured voltage(V)}} \times 1000$$

(5) After setting the parameter, command the spindle speed analog output as the spindle speed of the maximum voltage, again, and confirm the output voltage as 10V.

3731	Compensation value of the spindle speed analog output offset voltage (CSS)

『Modification authority』:Machine

『Parameter Type』: Word spindle

『Value Range』: -1000～+1000

『Default Setting』： 0

The parameter sets the compensation value of the spindle speed analog output offset voltage.

1. Set the standard setting value as 0.
2. Command the analog output voltage as 0V, which is the theoretical spindle speed.
3. Measure the output voltage.
4. Set the value in the following formula in parameter No.3731.

$$\text{setting value} = \frac{-8191 \times \text{offset voltage(V)}}{12.5}$$

5. After setting the parameter, command the analog output voltage as 0V, again, which is the theoretical spindle speed and confirm the voltage as 0V.

3740	Dwell time of the detection spindle speed reaching signal（SAD）

『Modification authority』:Machine

『Value Range』: 5～32767ms

『Default Setting』: 1000

Set the dwell time from executing S function to detecting the spindle speed reaching signal.

3741	Spindle maximum speed of gear 1 (MSG1)

3742	Spindle maximum speed of gear 2 (MSG2)

3743	Spindle maximum speed of gear 3 (MSG3)

3744	Spindle maximum speed of gear 4 (MSG4)

『Modification authority』:Machine

『Parameter Type』: Word spindle

『Default Setting』: 6000

『Value Range』: 0～32767r/min

The parameter sets the spindle maximum speed of each gear.

3770	Axis as the calculation reference during the constant surface speed control (ACS)

『Modification authority』:Machine

『Value Range』: 0～quantity of the control axes

『Default Setting』: 0

The parameter sets the axis as the calculation reference during the constant surface speed control.

Note: When it is set as 0, default X axis. Then, P value commanded in G96 block is not significant to the constant surface speed.

3771	Constant surface speed control mode (G96) spindle minimum speed (CFL)

『Modification authority』:Machine

『Value Range』: 0～32767r/min

『Default Setting』: 50

The parameter sets the spindle minimum speed when the constant surface speed control. During the constant surface speed control (G96), if the spindle speed is lower than the speed set by the parameter, it is limited in the parameter speed.

3772	Maximum spindle speed (MSS)

『Modification authority』:Machine

『Parameter Type』: Word spindle

『Value Range』: 0～32767r/min

『Default Setting』: 6000

The parameter sets the maximum spindle speed. The actual spindle speed is limited by the

maximum speed set by the parameter when the commanded spindle speed exceeds the maximum spindle speed, or the spindle speed after override exceeds the maximum spindle speed.

Note: 1. When the constant surface speed controls, no matter whether G96 or G97 is commanded, the spindle speed is limited by the maximum spindle speed.
2. When the setting value is 0, it is not limited by the speed.

3775	The default spindle in the multi-spindle selects the P command value (MPD)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～99

『Default Setting』: 0

In the multi-spindle control, when parameter MPP(NO.3703#3)=1 and MPA（NO.3706#2）=1; there is no specification for the P command value in the command S_ P_ after the power is turned on.

3781	In multi-spindle control, when code P is used for spindle selection (MPS)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～99

『Default Setting』: 0

When MPP(NO.3703#3)=1，In multi-spindle control, code P used for spindle selection is set with the parameter. And P code and S commands are specified in the same block

	#7	#6	#5	#4	#3	#2	#1	#0
4900								SFLR

『Modification authority』: Equipment management

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 SFLR The setting unit of parameters 4911 and 4912 during the FLR in the spindle speed fluctuation detection function.

0：1% is regarded as the unit

1：0.1% is regarded as the unit

4911	The allowable rate q of the spindle arrival commanded speed

	(SSQ)

『Modification authority』: Equipment management

『Way of Validating』:

『Value Range』:

『Default Setting』: 100

The allowable rate q of the spindle arrival commanded speed is set in the spindle speed changing detection function

4912	The rate r of spindle change without sending the spindle speed changing detection alarm (SSR)

『Modification authority』: Equipment management

『Way of Validating』:

『Value Range』:

『Default Setting』: 100

The rate r of spindle change is set without sending the alarm in the spindle speed change detection function.

4913	The change magnitude i of the spindle speed without sending the spindle speed change detection alarm (SSI)

『Modification authority』: Equipment management

『Way of Validating』:

『Value Range』: 0～99999

『Default Setting』: 100

The allowable magnitude i is set in the spindle speed change detection function without sending the alarm

4914	The time p from commanding the speed change to starting detecting the spindle speed change (SSP)

『Modification authority』: Equipment management

『Way of Validating』:

『Value Range』: 1～999999

『Default Setting』: 100

In the spindle speed change detection function, the time p from commanding the speed change to starting detecting the spindle speed change

Appendix 1.14 Parameter of Tool Compensation

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO		EVR				

『Modification authority』: Equipment management

『Default setting』: 0000 0000

#4 EVR In tool nose compensation mode C, when the tool compensation value is changed

0: It becomes valid from the next block which specifies T code.

1: It becomes valid from the next buffer block.

#6 EVO The rewritten value becomes valid when the compensation value of the tool position compensation mode is changed.

0: It is valid from the next block which specifies T code.

1: It is valid form the next buffer block.

	#7	#6	#5	#4	#3	#2	#1	#0
5002		LWM		LGT		LWT		LD1

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 LD1 Tool offset number

0: Specify through the last two digits of T code

1: Specify through the last one digit of T code

#2 LWT Tool wear compensation

0: Compensate through the tool traverse

1: Compensate through the coordinate system offset (there isn't any connection with LWM, and compensate in the block of T code)

#4 LGT Tool offset compensation mode

0: Compensate through the coordinate system offset (there isn't any connection with LWM, and compensate in the block of T code)

1: Compensate through the tool traverse

#6 LWM

0: Execute in T code block

1: Execute with axis movement meanwhile

Note: When LGT is 0, the offset is executed in T code block, and there isn't any connection with the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5003		LVC				CCN		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 CCN In the tool nose radius compensation mode, when the auto reference point return (G28) is commanded,

0: the tool nose traverses to the intermediate point.

1: But it is canceled until it traverses to the reference point.

#6 LVC Tool offset value is

0: Not cleared during resetting

1: Cleared during resetting

Note: The tooll offset function elimination by resetting should be enabled in the non-MDI mode.

	#7	#6	#5	#4	#3	#2	#1	#0
5004					TS1		ORC	

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#1 ORC Tool offset value

0: Specified by the diameter value (axes programmed by the diameter value)

1: Specified by the radius value

#3 TS1 The tool compensation value is directly input the touch inspection of sensor in the B function

0：It performs by 4 contactors

1：It performs by 1 contactor

	#7	#6	#5	#4	#3	#2	#1	#0
5005			QNI			PRC		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 PRC in direct input of tool offset compensation value and workpiece coordinate system offset amount, the PRC signal is

0: Used

1: Not used

#5 QNI The tool compensation measure value is directly input to the function B, the selection of the tool compensation number:

0：Operator selects by cursor

1：It performs by inputting the signal from PLC

	#7	#6	#5	#4	#3	#2	#1	#0
5006							TGC	OIM

『Modification authority』: Equipment management

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 OIM Switch between the inch system and the metric system, whether the tool offset value is auto changed

0: Not changed

1: Changed

#1 TGC Command T code in G50, G04 or G10 block

0: Not alarm

1: P/S alarms

	#7	#6	#5	#4	#3	#2	#1	#0
5008		CNS	CNF	MCR	CNV		CNC	CNI

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 CNI The tool nose radius compensation is interference checked

0: Execute

1: Not execute

#1 CNC When the tool nose radius compensation is interference checked and the difference between the programming movement direction and the offset movement direction is 90～270°

0: P/S alarms

1: Not alarm

#3 CNV The tool nose radius compensation (T serial) is interface checked and the vector is cleared

0: Execute

1: Not execute

#4 MCR If G41/G42 tool nose radius compensation is commanded in MDI mode, whether alarm

0: Not alarm

1: P/S alarm

Note: In MDI mode, the tool nose radius isn't compensated even it is set by the parameter.

#5 CNF When the tool nose radius compensation is interference checked, whether alarm when the internal full circle is cut

0: P/S alarms

1: Not alarm

#6 CNS The tool nose radius compensation is interference checked, whether alarm when the step is less than the tool radius

0: P/S alarms

1: Not alarm

	#7	#6	#5	#4	#3	#2	#1	#0
5009				**TSD**				**GSC**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 GSC The tool compensation measure value is directly input the offset write input signal in function B

0：It inputs from one side of machinery

1：It inputs from one side of PLC

#4 TSD The tool compensation measure value is directly input in function B, so that the movement direction distinguish specification is:

0：Disabled

1：Enabled

5010	**During the tool nose compensation, the limit value of the vector is ignored when the tool traverses along the corner outside (CLV)**

『Modification authority』: Equipment management

『Value Range』: 0～16383

SETTING UNITS	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

『Default Setting』: 0

The limit value of the minor traverse value is ignored when the tool nose radius compensation is set and the tool traverses along the corner outside.

5013	Maximum value of the tool wearing compensation value（MTW）

『Modification authority』: Equipment management

『Value Range』:

		IS-B	IS-C
SETTING UNITS	Input in metric system	0.001 mm	0.0001 mm
	Input in inch system	0.0001 inch	0.00001 inch
SETTING RANGE	Input in metric system	0～9 999 999	0～99 999 999
	Input in inch system		

『Default Setting』: 10

The parameter sets the maximum value of the tool wearing compensation value.

Note：When the set absolute value of the tool wearing compensation value exceeds the maximum value, it alarms: Input from MDI alarm: too many digits. Exceed range (XXXX——XXXX) (input range is in the bracket).
Input through G10 alarm: The offset value input by G10 is out of the specified range.

5015	In the manual tool measure, the distance (X1P) of the inspection sensor X+ contact surface

『Modification authority』: Equipment management

『Value Range』: -99999999～99999999

Set the record of each contact surface from measure reference position to inspection sensor. Specify the axis of diameter programming, setting value and diameter value.

5016	In the manual tool measure, the distance (X1M) of the inspection sensor X- contact surface

『Modification authority』: Equipment management

『Value Range』: -99999999～99999999

Set the record of each contact surface from measure reference position to inspection sensor. Specify the axis of diameter programming, setting value and diameter value.

5017	In the manual tool measure, the distance (Z1P) of the inspection sensor Z+ contact surface

『Modification authority』: Equipment management

『Value Range』: -99999999～99999999

Set the record of each contact surface from measure reference position to inspection sensor. Specify the axis of diameter programming, setting value and diameter value.

5018	In the manual tool measure, the distance (Z1M) of the inspection sensor Z- contact surface

『Modification authority』: Equipment management

『Value Range』: -99999999～99999999

Set the record of each contact surface from measure reference position to inspection sensor. Specify the axis of diameter programming, setting value and diameter value.

5020	Tool compensation measure value is directly input the tool offset number (TSB) in the function B

『Modification authority』: Equipment management

『Value Range』: 0～99

Set the tool offset number when the tool compensation value measure value is directly input to the function B (When the workpiece coordinate system offset value is set).

5021	In the manual tool measure, the memory movement interpolation cycle number before touching the detection sensor

『Modification authority』: Equipment management

『Value Range』: 0～8

Set the memorized movement interpolation cycle number for touching the inspection sensor, it is regarded as 8 when sets to 0.

5043	User the 1st offset axis number (YNSA1)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～6

『Default Setting』: 0

Set the axis number for compensating the tool offset value of the 1st offset axis, regardless of the 0.

5044	User the 2nd offset axis number (YNSA2)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～6

『Default Setting』: 0

Set the axis number for compensating the tool offset value of the 2nd offset axis, regardless of the 0.

5045	User the 3rd offset axis number (YNSA3)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～6

『Default Setting』: 0

Set the axis number for compensating the tool offset value of the 3rd offset axis, regardless of the 0.

5046	User the 4th offset axis number (YNSA)

『Modification authority』: System

『Validate method』: After power-on

『Value Range』: 0～6

『Default Setting』: 0

Set the axis number for compensating the tool offset value of the 4th offset axis, regardless of the 0.

Appendix 1.15 Parameter of Canned Cycle

The setting unit of canned cycle parameter is shown as follows:

	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

Appendix 1.15.1 Parameter of Canned Cycle

	#7	#6	#5	#4	#3	#2	#1	#0
5101						RTR		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 RTR In the G83 and G87

0：Specify the high-speed peck drilling cycle

1：Specify peck drilling cycle

	#7	#6	#5	#4	#3	#2	#1	#0
5102							MRC	

『Modification authority』:Equipment management

『Default Setting』: 0000 0000

#1 MRC The non-monotonic target shape is defined in multi-cycle command (G71 or G72),

or non-monotonic Z axis is in G73 cycle and the run-out value is in Z axis or the Finishing allowance X axis is non-monotonic

0: Not alarm

1: Alarm

	#7	#6	#5	#4	#3	#2	#1	#0
5104						**FCK**		

『Modification authority』:Equipment management

『Default Setting』:0000 0100

#2 FCK In combined canned cycles (G71, G72 and G73), the processing appearance is

0: Not checked

1: Checked

	#7	#6	#5	#4	#3	#2	#1	#0
5105						**RF2**		

『Modification authority』: Equipment management

『Default Setting』： 0000 0100

#2 RF2 In the type II of the canned cycle G71, whether perform the rough-machining cutting

0：YES

1：NO

5110	**M code locking C axis in the canned cycle of drilling holes（CMD）**

『Modification authority』:Equipment management

『Value Range』: 3～99

『Default Setting』:35

Set M code, which can lock C axis, during the canned cycle of drilling holes.

5114	**The return value in high-speed peck drilling cycle（HPDCRD）**

『Modification authority』: Equipment management

『Value Range』： 0～99 999 999×（system limit increase）

『Default Setting』： 1000

The return value in G83, G87 high-speed peck drilling cycle is set by the parameter.

5115	**The clearance value of peck drilling cycle（PDCRD）**

『Modification authority』: Equipment management

『Value Range』： 0～99 999 999×（system limit increase）

『Default Setting』：1000

The clearance value of G83, G87 peck drilling cycle is set by the parameter.

Appendix 1.15.2 Parameter of Thread Cutting Cycle

5130	Chamfering value of the thread cutting cycle（G76，G92） (THD)

『Modification authority』:Equipment management

『Value Range』: 0～99× (0.1 screw pitch)

『Default Setting』: 0

The parameter sets the beveling value of G76 and G92 thread cutting cycle.

5131	Chamfering angle in threading cycle(G92, G76) (CAT)

『Modification authority』:Equipment management

『Value range』:0～89

『Default』:0

The chamfering angle in threading cycle (G76) of the multiple repetitive canned cycle and the thread cutting cycle (G92) of single canned cycle are set by the parameter. When the parameter is set to 0, a value of 45 degree is determined.

Appendix 1.15.3 Parameter of Thread Cutting Cycle

5132	Cutting value of the combined canned cycle G71 and G72 (THC)

『Modification authority』:Equipment management

『Value Range』: 1～99 999 999

『Default Setting』: 1000

Set the cutting value of G71 and G72 combined canned cycle.

5133	Tool retraction amount of G71 and G72 combined canned cycle (MCE)

『Modification authority』:Equipment management

『Value Range』: 0～99 999 999

『Default Setting』: 0

Set the run-out value of G71 and G72 combined canned cycle.

5135	Tool retraction amount of G73 combined canned cycle along X axis direction (G73XE)

5136	Tool retraction amount of G73 combined canned cycle along Z axis direction (G73ZE)

『Modification authority』:Equipment management

『Value Range』: -99 999 999～99 999 999

『Default Setting』: 0

Set the run-out value of G73 combined canned cycle along with X and Z axes direction

5137	Partition times of G73 combined canned cycle (G73DC)

『Modification authority』:Equipment management

『Default Setting』: 1

『Value Range』: 1～999

Set the partition times of G73 combined canned cycle.

5139	Tool retraction amount of G74 and G75 combined canned cycles (G74G75R)

『Modification authority』:Equipment management

『Value Range』: 0～99 999 999

『Default Setting』: 1000

Set the reversal value of G74 and G75 combined canned cycle.

5140	Cut-in amount of G76 compound canned cycle (G76MID)

『Modification authority』:Equipment management

『Value Range』: 0～99 999 999

『Default Setting』: 0

Set the minimum cutting value of G76 combined canned cycle.

5141	Finishing allowance of G76 combined canned cycle (G76FA)

『Modification authority』:Equipment management

『Value Range』: 1～99 999 999

『Default Setting』: 500

Set the finishing allowance of G76 combined canned cycle.

5142	Finishing cycle times of G76 combined canned cycle (G76FC)

『Modification authority』:Equipment management

『Value Range』: 1～99

『Default Setting』: 1

Set the finishing cycle times of G76 combined canned cycle.

5143	Tool nose angle of G76 combined canned cycle (G76TNA)

『Modification authority』:Equipment management

『Value Range』: 0～99 (deg)

『Default Setting』: 60

Set the tool nose angle of G76 combined canned cycle.

5149	Override value for retraction in boring cycles (G85, G89) (BCRDOV)

『Modification authority』:Equipment management

『Value Range』： 0～2000

『Default Setting』： 200

Set the velocity override value (%) of the retraction operation in boring cycle, it is separately enabled to the feedrate. When this speed sets to 0, it equals to the 200% speed override.

Appendix 1.16 Parameter of Rigid Tapping

	#7	#6	#5	#4	#3	#2	#1	#0
5200	SRS	FHD	PCP	DOV		CRG		G84

『Modification authority』:Equipment management

『Default Setting』: 0000 0000

#0 G84 Method of commanding the rigid tapping

0: M code commands the rigid tapping before command G84/G88 (refer to parameter NO.5210).

1: M code doesn't command the rigid tapping. G84/G88 is taken as G code of the rigid tapping, and the common tapping is not used.

#2 CRG After the command of canceling the rigid tapping method, rigid tapping:

0: After the rigid tapping signal RGTAP changes to 0, the method is canceled.

1: Before the rigid tapping signal RGTAP changes to 0, the method is canceled.

#4 DOV Override during the rigid tapping run-out, in the tapping rigid, the override for drawing

0: Invalid

1: Valid, override value is set by parameter 5211

#5 PCP When address Q is commanded in tapping cycle/rigid tapping

0: Used as a high-speed peck tapping cycle

1: Used as a peck tapping cycle

#6 FHD Feed pause and single block running in rigid tapping is:

0: Forbidden

1: Allowed

#7 SRS To select a spindle used for rigid tapping in multi-spindle control:

0:The spindle selection signals SWS1～SWS3 are used

1:The rigid tapping spindle selection signals RGTSP1～RGTSP3

	#7	#6	#5	#4	#3	#2	#1	#0
5201				OV3	OVU	TDR		

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#2 TDR Cutting time constant in rigid tapping

0: Uses a same parameter NO.5261 during cutting and extraction

1: Not use a same parameter during cutting and extraction, parameter NO.5261 for cutting, parameter NO.5271 for extraction

#3 OVU The increment unit of the override parameter（№5211）is

0：1%

1：10%

#4 OV3 The spindle speed for tool extraction is specified by the program (address J). The override during the tool extraction is

0：Invalid

1：Valid

	#7	#6	#5	#4	#3	#2	#1	#0
5202		OVE						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 OVE The command range based on the extraction override command (address J) specified by the program during rigid tapping

0：100%～200

1：100%～2000%

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS				

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#4 OVS In rigid tapping, override by the feedrate override signal and invalidation of override by the override cancel signal is

0：Disabled

1：Enabled

Note1：When the feedrate override is set as valid, the extraction override is invalid。
Note2：The spindle speed override is fixed to 100%, irrelevant with the parameter。

	#7	#6	#5	#4	#3	#2	#1	#0
5209								RTX

『Modification authority』: Equipment management

『Default Setting』： 0000 0000

#0 RTX In rigid tapping, the drilling axis is

0：Selected by the plane

1：Fixed as Z axis by G84, X axis by G88

5210	M code commanding the rigid tapping（RTMC）

『Modification authority』:Equipment management

『Value Range』: 0～255

『Default Setting』: 0

M code is set to specify the rigid tapping method. When it is set as 0, CNC takes it as M29.

5211	Override of extraction during rigid tapping（RTEOV）

『Modification authority』:Equipment management

『Value Range』： 0～200

『Value Unit』： 1% or 10%

『Default Setting』： 100

The override value of extraction during rigid tapping。

Note 1：When parameter DOV(No.5200#4) is 1, the override value is valid.。
Note 2：When parameter OVU (No.5201#3) is 1, the unit of the setting data is 10%, and the override can be applied to the extraction of 2000%.

5213	Return or clearance in peck tapping cycle (PRTRD)

『Modification authority』: Equipment management

『Value Range』： 0～99999999

『Value Unit』：

SETTING UNITS	IS-B	IS-C	UNITS

linear axis（Input in metric system）	0.001	0.0001	mm
linear axis（Input in inch system）	0.0001	0.00001	Inch

『Default Setting』： 0

The return in high-speed peck tapping cycle or clearance in peck tapping cycle is set by the parameter.

5241 **Maximum spindle speed when rigid tapping（RTMS）**

『Modification authority』:Equipment management

『Value Range』: 0～9999

『Default Setting』: 1000

Set the spindle maximum speed in rigid tapping.

5261 **Time constant of linear acceleration/deceleration when rigid tapping（RTLT）**

『Modification authority』:Equipment management

『Value Range』: 0～4000ms

『Default Setting』:100

Time constant of linear acceleration or deceleration for the spindle for the rigid tapping.

5271 **Linear acceleration/deceleration time constant when rigid tapping retraction（RTET）**

『Modification authority』:Equipment management

『Value Range』: 0～4000ms

『Default Setting』: 100

Set the time constant of linear acceleration or deceleration of the spindle and the tapping axis during the rigid tapping run-out.

Note: The parameter is valid only when parameter TDR (NO.5201 BIT2) is set as 1.

5275 **Actually, the tapping axis lags behind the compensation cycle number (ZBK) sampled by spindle encoder in G84/G88**

『Modification authority』: Equipment management

『Value Range』： 0～10

『Default Setting』：6

Set in the G84/G88 common tapping (non-rigid tapping), the tapping axis lags behind the compensation cycle number sampled by spindle encoder. Generally, it is better set it to 4~8.

Appendix 1.17 Parameter of Polar coordinate interpolation

	#7	#6	#5	#4	#3	#2	#1	#0
5450							AFC	

『Modification authority』:Equipment management

『Default Setting』: 0000 0000

#0 AFC Whether use the auto override and the auto speed in the polar coordinate interpolation mode.

0: Not use

1: Use

Note: In the polar coordinate interpolation mode, the more closely the tool is near to the work piece center, the bigger the speed vector of the rotary axis is. If the center part exceeds the maximum cutting speed (parameter NO.5462), the servo (NO.411) alarms. Auto feedrate override and auto feedrate limit function auto controls the feedrate, then, the speed vector of the rotary axis doesn't exceed the maximum cutting feedrate.

5460	Specify the polar coordinate interpolation axis (linear axis)（LAI）
5461	Specify the polar coordinate interpolation axis (rotary axis)（RAI）

『Modification authority』:Machine

『Value Range』: 1～quantity of the control axes

『Default Setting』: NO.5460 is 1；NO.5461 is 5

Set the control axis numbers of the linear axis and the rotary axis for polar coordinate interpolation

5462	Maximum cutting feedrate of the polar coordinate interpolation（MFI）

『Modification authority』:Machine

『Default Setting』: 8000

	IS-B	IS-C	UNITS
Machine in metric system	0, 6～24 000	0, 6～10 000	mm/min
Machine in inch system	0, 6～9 600	0, 6～4 800	inch/min

Set the valid maximum feedrate of the polar coordinate interpolation. If the commanded speed is

greater than the value, the speed is limited by the maximum one. When the parameter is set as 0, the speed in the polar coordinate interpolation is limited by the maximum cutting feedrate (parameter NO.1422) value.

5463	**Allowable auto override percentage in polar coordinate interpolation （API）**

『Modification authority』:Equipment management

『Value Range』: 0～100 (%)

『Default Setting』: 0

When the polar coordinate interpolation is set, the percentages of the auto override are allowed to limit the cutting feedrate of the rotary axis.

The allowable speed of the rotary axis = Maximum cutting feedrate X override percentage

In polar coordinate interpolation, the more closely the tool is near to the work piece center, the bigger the speed vector of the rotary axis is. When it exceeds the allowable speed, the feedrate automatically multiplies by the override value calculated through the following formula:

Override = Allowable speed of the rotary axis/the speed vector of the rotary axis X 100%

If the revolving speed after timing the override still exceeds the allowable speed, the feedrate is limited in the allowable maximum cutting feedrate (auto speed limit function) .

Note: When the parameter value is set as 0, it is taken as 90%;

To limit the auto speed override and the auto speed, the parameter AFC (NO.5450#1) is set as 1.

Appendix 1.18 Parameter of User Macro Program

	#7	#6	#5	#4	#3	#2	#1	#0
6000			**SBM**					**G67**

『Modification authority』:Equipment management

『Default Setting』: 0000 0000

#0 G67 Macro program mode calling (G66) mode is not set, but mode calling command (G67) is canceled.

0: P/S alarms (NO.122)

1: Ignore G67

#5 SBM Whether use the single block to stop in the user macro program

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
6001	CLV	CCV						

『Modification authority』:Equipment management

『Default Setting』: 0100 0000

#6 CCV After reset, the user macro public variables 100～199 are：

0: Cleared as null

1: Not cleared

Note: In MDI mode, the macro public variables are not cleared after reset.

#7 CLV After resetting, the user macro program part vector 1～33 is

0: Cleared as null

1: Not cleared

	#7	#6	#5	#4	#3	#2	#1	#0
6004							MFZ	NAT

『Modification authority』:Equipment management

『Default Setting』: 0000 0000

#0 NAT The function command ATAN of the user macro program

0: Result of ATAN is 0～360.0 Result of ASIN is 270.0～0～90.0

1: Result of ATAN is -180.0～0～180.0n Result of ASIN is -90～0～90

#1 MFZ The angles of STN, COS or TAN, which are operation commands of the user macro program, are 1.0×10^{-8} or less, or the operation result is not exact 0

0: Underflow process

1: Reduction to 0

	#7	#6	#5	#4	#3	#2	#1	#0
6008		GMP	TMP					F0C

『Modification authority』: Equipment management

『Default Setting』： 0000 0000

#0 F0C The macro variable operation result

0: The alarm occurs when the data range exceeds ±1E308

1: The alarm occurs when the data range exceeds ±1E47

#5 TMP Whether allow the T code to call macro program

0：NO

1：YES

#6 GMP Whether allow M code calling the macro

0：No

1：Yes

6031	The beginning number of the variable to be protected in the common variables （#500~#999）（MPH）

『Modification authority』: Equipment management

『Value Range』： 500～999

『Default Setting』： 0

The beginning number of the variable in the common variables（#500~#999）is protected

6032	The end number of the variable to be protected in the common variables （#500~#999）（MPT）

『Modification authority』: Equipment management

『Value Range』： 500～999

『Default Setting』： 0

The end number of the variable in the common variables（#500~#999）is protected

6060	T code for calling Macro PROG. NO.9010 (TLM1)
6061	T code for calling Macro PROG. NO.9011 (TLM2)
6062	T code for calling Macro PROG. NO.9012(TLM3)
6063	T code for calling Macro PROG. NO.9013 (TLM4)
6064	T code for calling Macro PROG. NO.9014(TLM5)
6065	T code for calling Macro PROG. NO.9015 (TLM6)
6066	T code for calling Macro PROG. NO.9016 (TLM7)
6067	T code for calling Macro PROG. NO.90107(TLM8)
6068	T code for calling Macro PROG. NO.90108(TLM9)
6069	T code for calling Macro PROG. NO.90109(TLM10)

『Modification authority』: Equipment management

『Value Range』: 0～99999999

『Default Setting』: 0

T code for calling Macro PROG. NO.9010~9019 is set by the parameter.

6080	**M code for calling Macro PROG. NO.9020 (MLM1)**
6081	**M code for calling Macro PROG. NO.9021 (MLM2)**
6082	**M code for calling Macro PROG. NO.9022 (MLM3)**
6083	**M code for calling Macro PROG. NO.9023 (MLM4)**
6084	**M code for calling Macro PROG. NO.9024 (MLM5)**
6085	**M code for calling Macro PROG. NO.9025 (MLM6)**
6086	**M code for calling Macro PROG. NO.9026 (MLM7)**
6087	**M code for calling Macro PROG. NO.9027 (MLM8)**
6088	**M code for calling Macro PROG. NO.9028 (MLM9)**
6089	**M code for calling Macro PROG. NO.9029 (MLM10)**

『Modification authority』: Equipment management

『Value Range』: 3～99999999

『Default Setting』: 0

M code for calling Macro PROG. NO.9020~9029 is set by the parameter.

Appendix 1.19 Parameter of the Skip Function

	#7	#6	#5	#4	#3	#2	#1	#0
6200	**SKF**						**SK0**	

『Modification authority』: Machine

『Default Setting』: 0000 0000

#1 SK0 Set the valid state of the skip signal

0: valid when the input signal is “1”

1: valid when the input signal is “0”

#7 SKF Dry run and override for G31 jumping command are:

0: disabled

1: enabled

	#7	#6	#5	#4	#3	#2	#1	#0
6210		MDC						

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#6 MDC the measured automatic tool compensation value is

0: added to the current offset value

1: subtracted from the current offset value

	#7	#6	#5	#4	#3	#2	#1	#0
6240	IGA							AE0

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 AE0 Automatic tool compensation signal (X3.6), XAE2 (X3.7) indicates:

0: the measuring position is reached when it is 1

1: the measuring position is reached when it is 1

#7 IGA Automatic tool compensation function is:

0: used

1: not used

6241	Feedrate during automatic compensation (for XAE1 signal)(ATOF1)

6242	Feedrate during automatic compensation (for XAE2 signal)(ATOF2)

『Modification authority』: Machine

『Default Setting』: 1000

『Value setting』:

SETTIN UNIT	VALUE UNIT	VALID RANGE		DEFAULT
		IS-B	IS-C	
Metric	1mm/min	6～15000	6～12000	1000

Inch	0.1inch/min	6～6000	6～4800	

These two parameters set the feedrate during automatic tool compensation.

Note: When the setting value of parameter No. 6242 is valid, the setting value of parameter No. 6241 is valid too.

6251 **The γ value of X axis during automatic tool compensation (ATOR1)**

6252 **The γ value of Z axis during automatic tool compensation (ATOR2)**

『Modification authority』: Equipment management

『Value range』: 1～99999999

『Default Setting』: 1000

These two parameters set the γ value in tool compensation function in sequence.

Note: It is always set based upon the radius value regardless of the diameter or radius specification

6254 **The ε value of X axis during automatic tool compensation (ATOE1)**

6255 **The ε value of Z axis during automatic tool compensation (ATOE2)**

『Modification authority』: Equipment management

『Value range』: 1～99999999

SETTING UNIT	IS-B	IS-C	unit
Linear axis (metric input)	0.001	0.0001	mm
Linear axis (inch input)	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

These two parameters set the ε value in tool compensation function in sequence.

Note: The value is set in radius no matter diameter or radius programming is specified

Appendix 1.20 MPG Retraction Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6400		**MGO**						**RPO**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 RPO In the retraction function, the feedrate at the rapid traverse rate:

0：Clamped at the 10% of its equivalent override

1：Clamped at the 100% of its equivalent override

#6 MCO In the retraction function, perform the relative G code with measurement:

0：MPG pulse enabled

1：MPG pulse disabled, it always performs below the 100% override

	#7	#6	#5	#4	#3	#2	#1	#0
6401								**CRH**

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 CRH Whether forbid the MPG retraction in the hand MPG retraction method:

0：YES

1：NO

6405	**Clamp the override value (MLF) of the MPG retraction function at the rapid traverse rate**

『Modification authority』: Equipment management

『Value Range』: 0～100

『Default Setting』: 0

Set the override value for clamping at the MPG retraction function at the rapid traverse rate, when the 0 is set, this function is disabled, and the RPO(No.6400#0) is enabled (0～100).

6410	**The movement value of MPG per one pulse (MPM)**

『Modification authority』: Equipment management

『Value Range』: 0～100

『Default Setting』: 0

Set the movement value (0～100) of the MPG per one pulse by the override conversion

The mechanical movement value when actually rotates the MPG, which can be calculated according to the following method:

[Command speed] × [MPG override] × ([Parameter setting value]/100) ×8/60000 (mm or inch)

For example: The command speed is 30mm/min; the MPG override is 100; the movement value caused by MPG per one pulse in the case of the parameter No.6410 sets to 1, refer to the following formula:

[The movement value per one pulse]=30[mm/min] ×100×(1/100) ×(8/60000)[min]=0.004mm

Appendix 1.21 Parameter of Graphic Display

	#7	#6	#5	#4	#3	#2	#1	#0
6500					DPA			

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#3 DPA Current position display on the graphic display screen

0：Display the actual position including the tool nose radius compensation and tool offset

1: Display the programming position without tool compensation and offset

Appendix 1.22 Parameter of Run Hour and Parts Count Display

	#7	#6	#5	#4	#3	#2	#1	#0
6700							PRT	PCM

『Modification authority』: Equipment management

『Default Setting』: 0000 0000

#0 PCM M codes counting the total quantity of the processing parts and the quantity of the processing parts

0: M codes specified by M02 and M30 and parameter NO.6710

1: M codes only specified by parameter NO.6710

#1 PRT During setting, the signal PRTSF (F62.7) of the sufficient quantity of the processing parts is

0: Cut off

1: Not cut off

6710	**M codes counting the total quantity of the processing parts and the quantity of the processing parts (MPC)**

『Modification authority』: Machine

『Value Range』: 0～9999

『Default Setting』: 0

The machine program executes M codes set by the parameter, total quantity of the processing parts and quantity of the processing parts plus 1, respectively.

Note:When the setting value is 0, it is invalid (M00 can't count the parts). And it can't be set as 98 and 99, 198, neither.

『Modification authority』: Machine

『Value Range』: 0～9999

『Default Setting』: 0

When the quantity of the processing parts equals to that of the parts required being processed, the signal PRTSF (F62.7) of the enough quantity of the required parts outputs to PLC. However,

Note: If the quantity is 0, it is regarded as infinitely great, not output to PRTSF.

Appendix 1.23 Parameter for Tool Life Span Administration

	#7	#6	#5	#4	#3	#2	#1	#0
6800			GRC	GPS	SIG	LTM	GS2	GS1

『Modification authority』: Equipment management

『Validate method』: After power-on

『Default Setting』: 0000 00000

#0 GS1 The registered group numbers and the tool numbers of each 1 group can be changed by setting the parameters GS1, GS2 based upon the Max. group number in the parameter 6813.

#1 GS2 The registered group numbers and the tool numbers of each 1 group can be changed by setting the parameters GS1, GS2 based upon the Max. group number in the parameter 6813.

The relationships between GS1, GS2 and tool numbers are shown below:

GS2	GS1	Group Number	Tool Number
0	0	The 1/8 from the 1 to the Max. group number (No.6813)	1~16
0	1	The 1/4 from the 1 to the Max. group number (No.6813)	1~8
1	0	The 1/2 from the 1 to the Max. group number (No.6813)	1~4
1	1	The Max. group number (No.6813)	1~2

#2 LTM The specification of tool life span count type

0：Specify based upon the times

1：Specify based upon time

#3 SIG In the tool skip based on the signal, whether select the signal input group number by the tool group number

0：Do not input

1：Input

#4 GRS When inputting the tool-change resetting signal TLRST:

0: The clearing group is specified by the GRC of parameter 6800#5

1: Clear the registered executing data of the overall groups

#5 GRC When inputting the tool-change resetting signal TLRST, the specified group:

0: Automatically inspect the group used up of the life span by CNC

1: Select the signal specification by external tool group number

	#7	#6	#5	#4	#3	#2	#1	#0
6801						LVF	TSM	

『Factory type』： Equipment

『Modification authority』: Equipment management

『Default Setting』： 0000 00000

#1 TSM In the tool life span administration function, the life span count exists in the case of multi-offset command

0: The counting is performed based upon the each same tool number

1: The counting is performed based upon the each cutter

#2 LVF Use the time count life span value in the tool life span administration function, the tool life span count override signal *TLVO～*TLV9<G049.0～G050.1> places at:

0：Disabled

1：Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
6802	RMT							T99

『Default Setting』： 0000 00000

『Modification authority』: Equipment management

『Validate method』：Immediately

#0 T99 When the tool group of the life span is used up, perform the M99 in the main program:

0: Do not output the tool-change signal

1: Output the tool-change signal, and then enter to the auto operation stop state.

#7 RMT Tool life span predicted signal TLCHB

0: The residual value of life-span (life-span value — life-span counter), ≤ the remainder value of the ON life-span when resetting the counting value > It is OFF when resetting the counting value

1: The surplus of life-span = ON during resetting counting value, the residual value of life-span ≠ OFF during the resetting counting value

	#7	#6	#5	#4	#3	#2	#1	#0
6804		LFI						

『Default Setting』： 0000 00000

『Modification authority』: Equipment management

『Validate method』：Immediately

#6 LFI The selected tool life-span counting in the tool life administration

0：Enabled

1: Count the disabled signal LFCIV (G48.2) by tool life-span, the shifting is performed between enabled or disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
6805							FGL	

『Default Setting』： 0000 00000

『Modification authority』: Equipment management

『Validate method』：Immediately

#1 FGL life-span counting type is registered based upon the life data of G10 in the case of the specified time

0: Unit by 1 minute

1: Unit by 0.1 second

6810	Tool life-span administration ignore number (TLC)

『Default Setting』： 0

『Modification authority』: Equipment management

『Value Range』： 0～9999 9999

『Validate method』：Immediately

When the figure exceeds the set value by using the T code, some value deducted from the set value based upon the T code numerical value becomes the tool group number of the tool life-span administration.

6811	The M code is used by tool life-span counting restart (MRN)

『Default Setting』： 0

『Modification authority』: Equipment management

『Value Range』： 0～127

『Validate method』：Immediately

In this case, the life-span existence is set by times, the tool group when the tool life-span counting restarting specifies the M code is used up.

The tool-change signal (TLCH) may also be output even if only one signal; when it is set to 0, the parameter will then be ignored.

6813	The Max. group number of the tool life-span administration (MTN)

『Default Setting』： 0

『Modification authority』: Equipment management

『Value Range』： 0, 8, 16, 32, 64, 128

『Validate method』: After power-on

Set the used top group number of each path, after this parameter is set, the power should be temporarily turned off.

6844	The residual span using times of tool (TLP)

『Default Setting』： 0

『Modification authority』: Equipment management

『Value Range』：0~65535

『Validate method』：Immediately

In the case of the tool life-span is specified, cutter output span reaches to the tool residual span of the predictive signal (Using times).

6845	The remainder span using time of tool (TLR)

『Default Setting』：0

『Modification authority』: Equipment management

『Value Range』： 0~4300

『Validate method』：Immediately

In the case of the tool life-span is specified, cutter output span reaches to the tool residual span of the predictive signal (Using time).

Appendix 1.24 Parameter of MPG Feed

	#7	#6	#5	#4	#3	#2	#1	#0
7100				HPF				JHD

『Modification authority』: Machine

『Default Setting』: 0000 0000

#0 JHD MPG feeding in JOG mode or increment feeding in MPG feed mode

0: Invalid

1: Valid

	JHD=0		JHD=1	
	JOG MODE	MPG MODE	JOG MODE	MPG MODE
JOG feeding	O	×	O	×

MPG feeding	×	O	O	O
Increment feeding	×	×	×	O

#4 HPF When MPG feedrate exceeds the manual rapid movement speed

0: The speed is limited in the manual rapid movement speed, the pulse exceeding the manual rapid movement part is ignored (The scale of MPG does not comply with the movement amount)

1: The speed is limited in the manual rapid movement speed; the exceeding part isn't ignored but saved in CNC. (Although MPG is stopped, the machine still moves the pulse value saved in CNC and then stops.)

	#7	#6	#5	#4	#3	#2	#1	#0
7102								**HNGx**

『Modification authority』: Machine

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 HNGx Revolving direction of each axis movement direction and that of MPG

0: Same

1: Opposite

	#7	#6	#5	#4	#3	#2	#1	#0
7103						**HNT**		

『Modification authority』: Machine

『Default setting』: 0000 0000

#2 HNT Movement amount override of the incremental feed/MPG feed is set to the one that is selected by the MPG feed movement selection signal

0：1

1：10 times

7110	**Number of MPG (NMP)**

『Modification authority』：Machine

『Value Range』: 1~2

『Default Setting』: 1

Set the quantity of MPG.

7113	**MPG feed override M(MFM)**

『Modification authority』：Machine

『Value Range』: 1～127

『Default Setting』: 100

Set the override when MPG feeding movement value selection signals MP1=0, MP2=1.

7114	MPG feed override N(MFN)

『Modification authority』: Machine

『Value Range』: 1～1000

『Default Setting』: 1000

Set MPG feeding override when MPG feeding movement value selecting signals MP1=1, MP2=1.

MOVEMENT VALUE SELECTING SIGNAL		MOVEMENT VALUE (MPG FEEDING)
MP2	MP1	
0	0	Minimum setting unit * 1
0	1	Minimum setting unit * 10
1	0	Minimum setting unit * M
1	1	Minimum setting unit * N

7117	Allowable pulse cumulative value in MPG feed (APM)

『Modification authority』: Machine

『Value Range』: 0～1000

『Default Setting』: 1000

When MPG feeding instance exceeds the rapid movement speed, the pulse exceeding the rapid movement is not canceled but saved. The parameter sets the allowable value of the memory capacity.

Note: When overrides, such as X100 or more than it, are selected, MPG rapidly turns round. MPG feeding is more than the rapid movement speed; the speed is limited by the rapid movement speed. The pulse exceeding the rapid movement speed is ignored; therefore, the scale value of MPG doesn't comply with the actual movement value. Then, If the allowable value is preset in the parameter, the pulse exceeding the rapid movement speed is not canceled, but saved in CNC temporarily (the part exceeding the allowable value is ignored). When MPG revolving speed becomes slower or the revolving stops, the saved pulse changes into the movement command and outputs. Pay attention to it if the allowable value is set too big, even MPG is stopped revolving, CNC won't stop until the remaining pulse is completed.

Appendix 1.25 Parameters of Program Restart

	#7	#6	#5	#4	#3	#2	#1	#0
7300	MOU	MOA						

『Modification authority』：Machine

『Default Setting』: 0000 0000

#6 MOA In program restart operation, before movement to a machining restart point

0:The last M, S, T and B codes are output

1:All M codes and the last S, T and B codes are output

#7 MOU In program restart operation, before movement to a machining restart point after restart block search

0：The M, S, T and B codes are NOT output

1：The M, S, T and B codes are output

	#7	#6	#5	#4	#3	#2	#1	#0
7301								ROF

『Modification authority』：Machine

『Default Setting』: 0000 0000

#0 ROF In the restart coordinate display on the program restart screen, whether display the various tool compensation values

0：Display the tool compensation and offset

1：Set by DAL, bit 6 of parameter No.3104 or DAC, bit 7 of parameter No.3104

7310	The axis sequence by dry run after a program is restarted (ROAX)

『Modification authority』：Machine

『Value Range』: 1～quantity of the control axes

『Default Setting』: 1

The axis sequence when the machine moves to the restart point by dry run and is specified by the dedicated axis after a program is restarted

Appendix 1.26 Polygon Machining Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7603		PQS		PSM	PLR			

『Modification authority』：Machine

『Default Setting』: 0000 0000

#3 PLR The tool rotation axis with each movement value of the polygon machining

0: Round off by the setting value of the parameter 7620

1: Round off based upon 360

#4 PSM The workpiece rotation axis working mode of the polygon machining

0: Speed mode

1: Position mode

#6 PQS The PQ value of the polygon machining is:

0: The rotation ratio between the tool rotation axis and workpiece rotation axis

1: The ratio value between the polygon number and tool number

7610	The controllable axis number (PCA) for using the tool rotation axis of the polygon machining

『Modification authority』: Machine

『Value Range』: 0～quantity of the control axes

『Default Setting』: 0

Set the controllable axis number of the tool rotation axis for using the polygon machining, when it is set to 0, which means that this function does not work.

7620	The movement amount (PEM) per each rotation for using tool rotation axis of polygon machining

『Modification authority』: Machine

『Value Range』: 0～3600000

『Default Setting』: 0

Set the movement amount per each cycle of the tool rotation axis

7621	The upper-limit speed (PSM) for using the tool rotation axis of the polygon machining

『Modification authority』: Machine

『Value Range』: 0～99999999

『Default Setting』: 0

Set the upper-limit speed of the tool rotation axis

Appendix 1.27 Parameter of PLC Axis Control

	#7	#6	#5	#4	#3	#2	#1	#0
8001		AUX	NCC		RDE	OVE		MLE

『Modification authority』：Machine

『Default Setting』: 0000 0000

#0 MLE Whether the locking machine signal MLK of PLC control axis is valid

0: Valid

1: Invalid

#2 OVE Signals relative with the dry run and the override controlled by PLC axis

0: Same signals controlled by CNC

1: Signals especially used in PLC

#3 RDE In PLC axes control, whether the dry run is valid for the rapid feeding commands

0: Invalid

1: Valid

#5 NCC For PLC control axes (the control axes select the axes chosen by the signal), command the program to command the movement

0: According to the axis control command, PLC controls the axis, P/S (No.139) alarms; the axis is not controlled, CNC command is valid.

1: P/S (No.139) alarms.

#6 AUX The number of bytes for the code of an auxiliary function to be output is

0：One

1：Two

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10		DWE	RPD

『Modification authority』：Machine

『Default Setting』: 0000 0000

#0 RPD The rapid movement speed of PLC control axis

0: Feedrate set by parameter No.1420

1: In axis control command, feedrate set by feedrate Value

#1 DWE When use the increment system IS-C, the minimum time specified by the pause command during PLC axis control

0: 1ms

1: 0.1ms

#3 F10 In PLC axis control, the minimum increment units of the cutting feedrate (per min)

F10	Input in metric system	Input in inch system
0	1mm/min	0.01inch/min
1	10mm/min	0.1inch/min

#4, #5 PR1, PR2 In PLC axis control, the least increment unit of cutting feed

PF2	PF1	Speed
0	0	1/1
0	1	1/10
1	0	1/100
1	1	1/1000

#6, #7 FR1, FR2 The feedrate units of per revolution feeding during PLC axis control

FR2	FR1	Input in metric system	Input in inch system
0	0	0.0001mm/rev	0.000001inch/rev
1	1		
0	1	0.001mm/rev	0.00001inch/rev
1	0	0.01mm/rev	0.0001inch/rev

	#7	#6	#5	#4	#3	#2	#1	#0
8004		**NCI**	**DSL**			**JFM**		

『Modification authority』: Machine

『Default Setting』: 0000 0000

#2 JFM Feedrate units of continuous feeding (06h) of PLC control axis

INCREMENT SYSTEM	JFM	INPUT IN METRIC SYSTEM	INPUT IN INCH SYSTEM	ROTARY AXIS
IS-B	0	1mm/min	0.01inch/min	1deg/min
	1	200mm/min	2.00inch/min	200deg/min
IS-C	0	0.1mm/min	0.001inch/min	0.1deg/min
	1	20mm/min	0.200inch/min	20deg/min

#5 2DSL When selecting the axes controlled by PLC is forbidden, if the axes are tried to exchange

0: Failed and P/S No.139 alarms

1: Axes, without commanding the channel, are executed exchanging

#6 NCI During decelerating the axes controlled by PLC, in-position check is

0: Executed

1: Not executed

	#7	#6	#5	#4	#3	#2	#1	#0
8005							CDI	

『Modification authority』：Machine

『Default Setting』: 0000 0000

#1 CDI When PLC control axis selects the diameter programming, under PLC axis control

0: Radius programming specifies the movement distance

1: The diameter programming specifies the movement distance

#2 R10 When the RPD parameter(No.8002#0) is set to "1", the unit for specifying a rapid traverse rate for the PLC axis is

0：×1

1：×10

	#7	#6	#5	#4	#3	#2	#1	#0
8006	EAL			EFD				

『Modification authority』：Machine

『Default Setting』: 0000 0000

#4 EFD In axis control by PLC, the unit for specifying feed cutting for PLC axis is

0：×1

1：×100

#7 EAL In axis control by PLC, the function that allows the alarm signal to be reset by a CNC reset operation is

0: Not release the alarm of PLC control axis

1: Release the alarm of PLC control axis

8010	Selecting each axis DI/DO group controlled by PLC（EPAS）

『Modification authority』：Machine

『Parameter Type』: Word axis type

『Value Range』: 0～4

『Default Setting』: 0

Each DI/DO group controlled by each PLC axis, which is shown as the following list:

NUMERICAL VALUE	REMARK
0	The axis is not controlled by PLC
1	DI/DO in group A is used
2	DI/DO in group B is used
3	DI/DO in group C is used

4	DI/DO in group D is used

8022	**Maximum feedrate of feeding/per revolution controlled by PLC axis（EPMF）**

『Modification authority』：Machine

『Parameter Type』: Word axis type

『Value Range』:

INCREMENT SYSTEM	VALUE UNITS	VALID VALUE RANGE	
		IS-B	IS-C
Machine in metric system	1mm/min	6~15000	6~12000
Machine in inch system	0.1inch/min	6~6000	6~6000
Rotary axis	1deg/min	6~15000	6~12000

『Default Setting』: 6

Set the maximum feedrate of feeding/per revolution controlled by PLC axis.

8028	**For each PLC control axis, the linear acceleration or deceleration time constant specified by speed command during JOG feeding （EPAT）**

『Modification authority』：Machine

『Parameter Type』: Word axis type

『Value Range』: 0~3000ms

『Default Setting』: 100

Specify the linear acceleration or deceleration time constant during JOG feeding

Note: If it is set to "0", the system doesn't control the acceleration and deceleration.

8030	**Shift of reference position for PLC controlled axes (RPS)**

『Modification authority』：Machine

『Parameter Type』: Word axis type

『Value Range』: -99999999～99999999

『Default Setting』: 0

Set the shift of reference position for PLC controlled axes

Appendix 1.28 Parameter of the Basic Function

8130	Total quantity of the controlled axes (TCA)

『Modification authority』：System

『Validate method』: After power-on

『Value Range』：2～6

『Default Setting』：2

Set the total quantity of the axes controlled by CNC system.

	#7	#6	#5	#4	#3	#2	#1	#0
8131								HPG

『Modification authority』：Machine

『Validate method』: After power-on

『Default Setting』: 0000 0001

#0 HPG Whether use MPG feeding

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
8132							YOF	TLF

『Modification authority』：Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 TLF Whether use the tool work life management function

0: Not use

1: Use

#1 YOF The Y-axis offset is

0: Not used

1: Used

	#7	#6	#5	#4	#3	#2	#1	#0
8133						SCS		SSC

『Modification authority』：Machine

『Validate method』: After power-on

『Default Setting』: 0000 0001

#0 SSC Whether use the function of the constant surface speed (G96)control

0: Not use

1: Use

#2 SCS Whether use CS outline control function

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
8134						CCR		

『Modification authority』：Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#2 CCR The chamfering/corner R is

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
8135	RPTH					NSQ		

『Modification authority』：Machine

『Validate method』: After power-on

『Default Setting』: 0000 0100

#2 CCR The program restarting function is

0：Used

1：Not used

#7 RPTH Whether use the thread recovery function

0：NO

1：YES

	#7	#6	#5	#4	#3	#2	#1	#0
8136					NOP			

『Modification authority』：Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#3 NOP Whether use the soft machine tool panel

0：NO

1：YES

Appendix 1.29 Parameter for Slopping Axis Control

	#7	#6	#5	#4	#3	#2	#1	#0
8200						AZR		AAC

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 AAC Whether perform the slopping axis control

0：NO

1：YES

#2 AZR When performing the slopping axis manual reference point return in its axis control method

0：Perpendicular axis is also moved at the same time

1：Perpendicular axis does not move

	#7	#6	#5	#4	#3	#2	#1	#0
8209								ARF

『Modification authority』: Machine

『Validate method』: After power-on

『Default Setting』: 0000 0000

#0 ARF Move to the reference point from the intermediate pont specified by G28/G30 based upon the slopping axis control:

0：The motion of the slopping coordinate system

1：The motion of Cartesian coordinate system

8210	**The slopping angle (INA) in the slopping axis control**

『Modification authority』: Machine

『Validate method』: After power-on

『Value Range』: -1800000~1800000

『Default Setting』: 0

This parameter sets the slopping axis angle in its axis control

Setting unit: IS-B 0.001deg; IS-C 0.0001deg.

8211	**The slopping axis number (ANS) for performing the slopping axis control**

8212	The rectangular axis number (ANC) for performing the slopping axis control

『Modification authority』：Machine

『Validate method』: After power-on

『Value Range』: 0~6

『Default Setting』: 0

This parameter sets the slopping axis number when the slopping axis is controlled. When one of any parameters sets to 0, alternatively, either the same numbers are set or non-control axis number is set, which means that the function is disabled.

Appendix 1.30 Parameter of GSKLink Communication Function

	#7	#6	#5	#4	#3	#2	#1	#0
9000							GCRC	GNET

『Modification authority』：Machine

『Validate method』: After power-on

『Default Setting』: 0

#0 GNET Whether the system GSKLink communication function is enabled

0：NO

1：YES

#1 GCRC Whether the system GSKLink communication data is performed the verification

0：NO

1：YES

Appendix 2 Standard PLC Function Configuration

Appendix 2.1 Standard Panel on the Machine Tool

Appendix 2.1.1 GSK988TA1 Standard Panel on Machine Tool

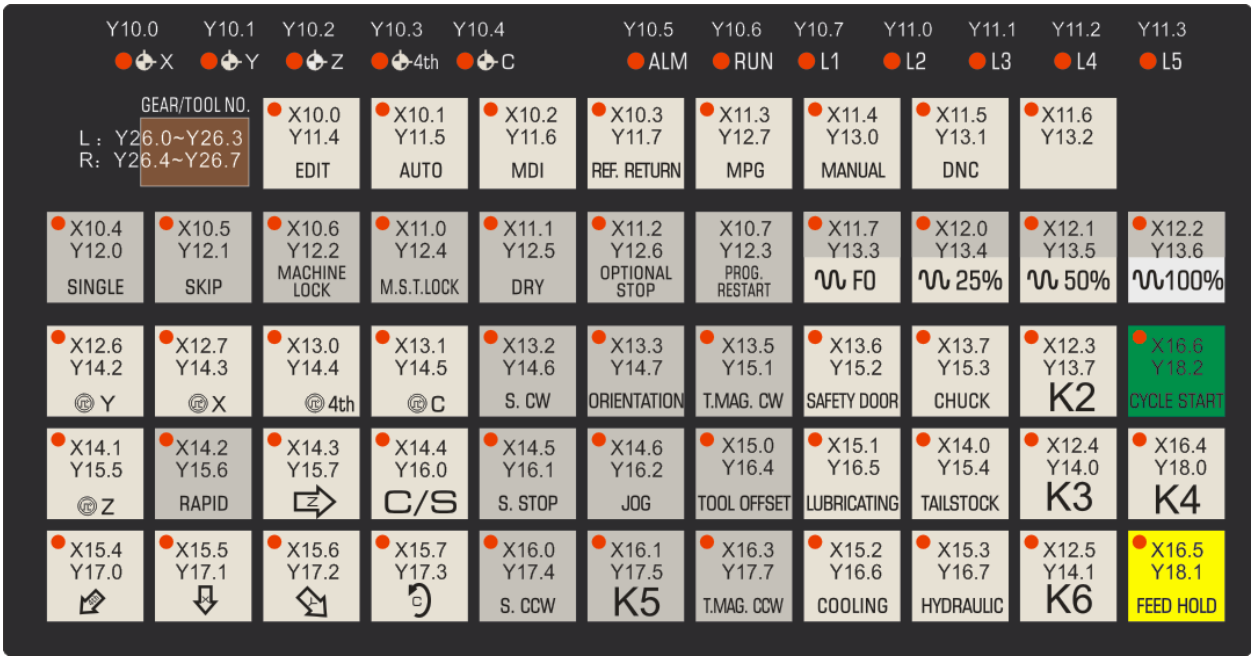


Fig 2-1 GSK988TA1 Standard layout of operation panel

Note: It is the same size between GSK988TA1-H and GSK988TA1 about the address of Standard Panel

Appendix 2.1.2 GSK988TA Standard Panel on Machine Tool

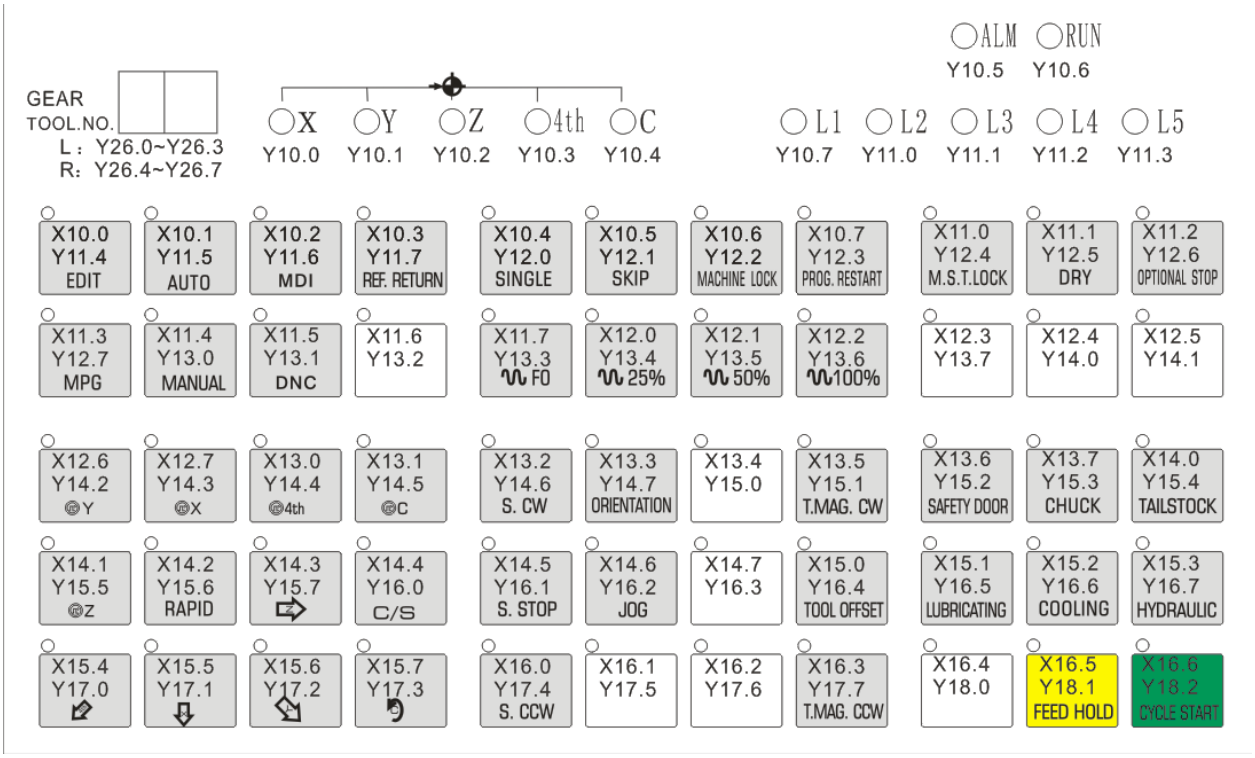


Fig.2-2 GSK988TA Standard layout of operation panel

Appendix 2.1.3 GSK988TA-H Standard Panel on Machine Tool

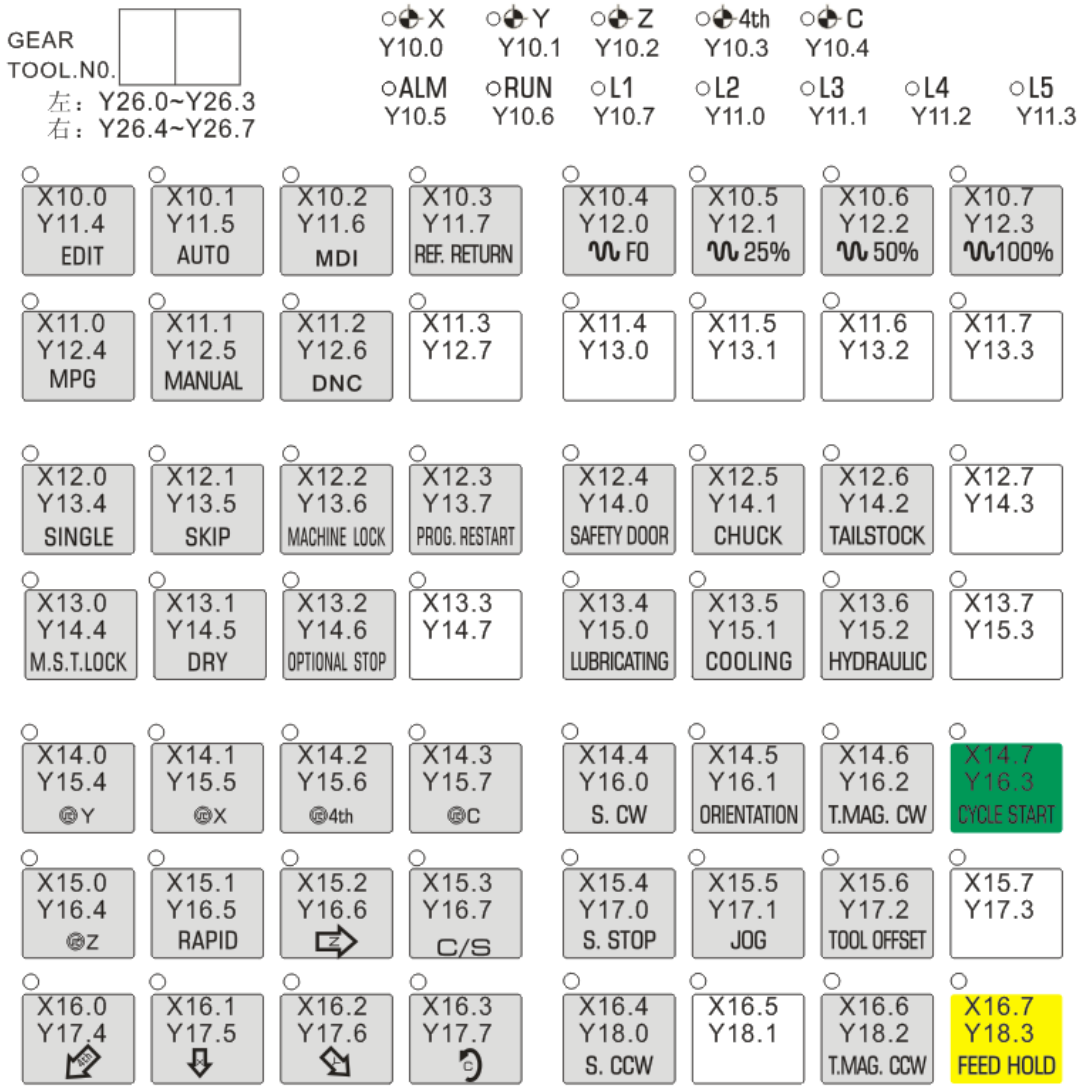


Fig. 2-3 GSK988TA-H Standard layout of operation panel

Appendix 2.1.4 GSK988TB Standard Panel on the Machine Tool

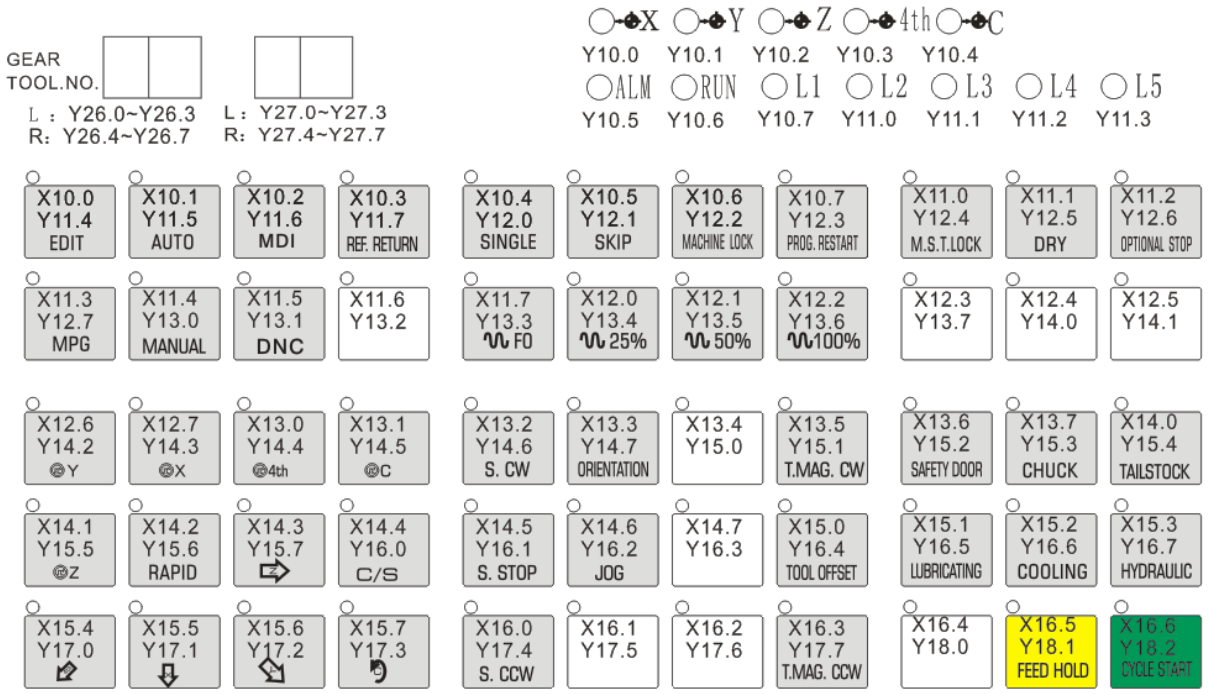


Fig. 2-4 988TB Standard layout of operation panel

Note: It is the same size between GSK988TB-H and GSK988TB about the address of Standard Panel

Appendix

Appendix 2.2 Definitions of X and Y Addresses of the Ladder Diagram

I/O of GSK988TA/988TA1/988TB is classified into high speed I/O signal and the common I/O one. The high speed I/O signals are those of CN61 on CNC back cover. The common I/O signal is the extension signals of the remote I/O unit. The function of I/O signal of CNC (except for the signal of the marked fixed address) is defined by the system internal PLC program (the ladder diagram). When GSK988TA/988TA1/988TB turning machine CNC is configurated with the machine, I/O function is set by the machine manufacturer; please refer to the user manual for the machine manufacturer about the details.

In this chapter , please pay attention that the common I/O singals（X and Y addresses）function is mainly for the starndard PLC program of GSK988TA/988TA1/988TB.

Appendix 2.2.1 High speed I/O interface

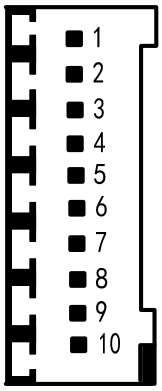


Fig. 2-5 Pins of CN61

Corresponding connector pins	PLC address	Function defined by the standard PLC addresses		Note
CN61.1	0V			
CN61.2	X0.0		Reserved	
CN61.3	X0.1		Reserved	
CN61.4	X0.2		Reserved	
CN61.5	X0.3		Reserved	
CN61.6	X0.4	SKIP	G31 skip signal	
CN61.7	X0.5	ESP	Emergency stop input signal	
CN61.8	X0.6	G36	G36 skip signal	
CN61.9	X0.7	G37	G37 skip signal	
CN61.10	0V			

Appendix 2.2.2 Common machine I/O interface

The all-purpose output/input of the GSK988TA/988TA1/988TB is distributed by connecting the I/O unit with the GSKLink. Up to 4 I/O units can be connected by GSKLink, up to 48 input points and 32 output points of each I/O unit, and the address use range is X80~X127 and Y80~Y127 of which the user should configure the system address by herself/himself.

An I/O unit with 48 input points and 32 output points is configured in the standard configuration of the GSK988TA/988TA1/988TB system. The configuration address of standard ladder diagram in the system is X100~X105 and Y100~Y103. The overall I/O introduced in this User Manual, however, if the difference condition occurs, refer to the User Manual offered by machine tool manufacturer. The signal addresses are subject to it.

PLC address	Function defined by standard PLC address		Remark
X100.0	SAGT	Protection door detection signal	
X100.1		Reserved	
X100.2	DIQP	Chuck input signal	
X100.3		The 1st axis deceleration signal	
X100.4	DITW	Tailstock control signal	
X100.5		Emergency stop input signal	
X100.6	PRES	Pressure detection signal	
X100.7	T05	Tool position signal 5/ tool post pre-indexing signal (Yantai AK31)/Sensor E (Liuxin Tool Post)	
X101.0	T06	Tool position signal 6/ tool post pre-indexing signal (Yantai AK31)/Sensor F (Liuxin Tool Post)	
X101.1	T07	Tool position signal 7/ tool post overheat signal (Yantai AK31)	
X101.2	T08	Tool position signal 8	
X101.3		Reserved	
X101.4		Reserved	
X101.5	M41I	The 1st gear stage in-position	
X101.6	M42I	The 2nd gear stage in-position	
X101.7	T01	Tool position signal 1/T1 (Yantai AK31)/Sensor A (Liuxin Tool Post)	
X102.0	T02	Tool position signal 2/T2 (Yantai AK31)/ Sensor B (Liuxin Tool Post) Sensor A (Liuxin Tool Post)	
X102.1	T03	Tool position signal 3/T3 (Yantai AK31)/Sensor C (Liuxin Tool Post)	
X102.2	T04	Tool position signal 4/T4 (Yantai AK31)/Sensor D (Liuxin Tool Post)	
X102.3		Reserved	
X102.4		Reserved	
X102.5		Reserved	
X102.6	TCP	Tool post lock signal Tool post proximity switch signal (Yantai AK31)	
X102.7		Reserved	

PLC address	Function defined by standard PLC address		Remark
X103.0	LMI1+	The 1st axis + side overtravel signal	
X103.1	LMI2+	The 2nd axis + side overtravel signal	
X103.2	LMI3+	The 3rd axis + side overtravel signal	
X103.3	WQPJ	Chuck in-position signal (outer chuck clamping and inner chuck unclamping)	
X103.4	NQPJ	Chuck in-position signal (inner chuck clamping and outer chuck unclamping)	
X103.5		Reserved	
X103.6		Reserved	
X103.7		Reserved	
X104.0	LMI1-	The 1st axis – direction overtravel signal	
X104.1	LMI2-	The 2nd axis – direction overtravel signal	
X104.2	LMI3-	The 3rd axis – direction overtravel signal	
X104.3	LMI4+	The 4th axis + direction overtravel signal	
X104.4	LMI4-	The 4th axis - direction overtravel signal	
X104.5	LMI5+	The 5th axis + direction overtravel signal	
X104.6	LMI5-	The 5th axis - direction overtravel signal	
X104.7		Reserved	
X105.0		Reserved	
X105.1		Reserved	
X105.2		Reserved	
X105.3		Reserved	
X105.4		Reserved	
X105.5		Reserved	
X105.6		Reserved	
X105.7		Reserved	
Y100.0	M08	Cooling output signal	
Y100.1	M32	Lubrication output signal	
Y100.2		the hydraulic station output signal	
Y100.3	M03	Spindle CCW signal	
Y100.4	M04	Spindle CW signal	
Y100.5	M05	Spindle stop signal	

PLC address	Function defined by standard PLC address		Remark
Y100.6	M35	Spindle hold output signal	
Y100.7	SPZD	Spindle braking output signal	
Y101.0	M41	Spindle gear 1 output signal	
Y101.1	M42	Spindle gear 2 output signal	
Y101.2	M43	Spindle gear 3 output signal	
Y101.3	M44	Spindle gear 4 output signal	
Y101.4	M12(DOQPJ)	Outer chuck clamping output / Inner chuck unclamping output signal	
Y101.5	M13(DOQPS)	Outer chuck unclamping output /inner chuck clamping output signal	
Y101.6	TL+	Tool post forward rotation output signal	
Y101.7	TL-	Tool post reverse rotation output signal	
Y102.0		Tool post motor braking signal (Yantai AK31)/ tool post unclamping output (Liuxin Tool Post)	
Y102.1		Tool post pre-indexing electromagnet signal (Yantai AK31)/ Tool post lock output (Liuxin Tool Post)	
Y102.2	YLAMP	Tri-colored lamp – yellow (normal state, non-running, non-alarm)	
Y102.3	GLAMP	Tri-colored lamp – green (running state)	
Y102.4	RLAMP	Tri-colored lamp – red (alarm state)	
Y102.5	M10	Tailstock advancing output signal	
Y102.6	M11	Tailstock retracting output signal	
Y102.7		Reserved	
Y103.0	M37	Chip	
Y103.1	M38	Chip	
Y103.2		Reserved	
Y103.3		Reserved	
Y103.4	SORI	Spindle orientation signal	
Y103.5	SEC0	Spindle orientation selection signal 1	
Y103.6	SEC1	Spindle orientation selection signal 2	
Y103.7	SEC2	Spindle orientation selection signal 3	

Note 1: The addresses of X100.0~X105.7 are the high level input valid, that is to say, when the input signal is connected with +24V, X address signal status is 1, otherwise, the status is 0.

Note 2: When Y address signal status output by low level is 1, the output signal is connected with 0V (0V output); when Y address signal status is 0, the output signal is high resistance.

Appendix 2.2.3 Interface of the Handhold Box

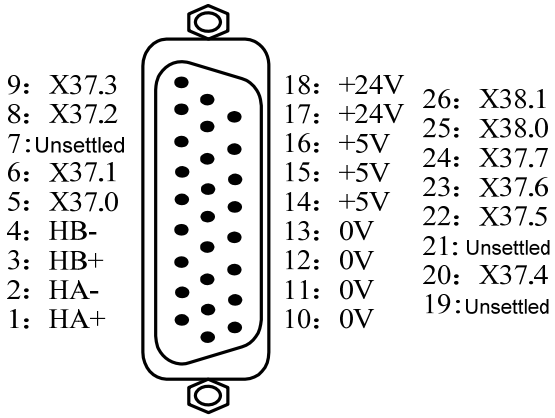


Fig.2-6 CN32 MPG interface

（Pin socket of type D in 26 cord）

DB Pin	Signal Definition	Signal Instruction	Function defined by standard PLC address
CN32.1,CN32.2	HA+，HA-	MPG phase A signal input	/
CN32.3,CN32.4	HB+，HB-	MPG phase B signal input	/
CN32.5	X37.0	PLC signal address, switch amount input	External MPG box X axis selection signal
CN32.6	X37.1	PLC signal address, switch amount input	External MPG box Y axis selection signal
CN32.8	X37.2	PLC signal address, switch amount input	External MPG box Z axis selection signal
CN32.9	X37.3	PLC signal address, switch amount input	External MPG box ×1 gear signal
CN32.22	X37.5	PLC signal address, switch amount input	External MPG box ×10 gear signal
CN32.23	X37.6	PLC signal address, switch amount input	External MPG box ×100 gear signal
CN32.24	X37.7	PLC signal address, switch amount input	External MPG box ×X1000 gear signal
CN32.25	X38.0	PLC signal address, switch amount input	External MPG box the 4th axis selection signal
CN32.26	X38.1	PLC signal address	External MPG box the 5th axis selection signal
CN32.20	X37.4	PLC signal address	External MPG box the 6ix axis selection signal
CN32.10，CN32.11 CN32.12，CN32.13	0V	0V	/
CN32.14，CN32.15 CN32.16	+5V	+5V	/
CN32.17,CN32.18	+24V	+24V	/

Note: When X37.0~X38..0 as high level input are valid, that is to say, when the input signal is connected with +24V, the input is valid, and X address status is 1;

Appexdix 3 Alarm Troubleshooting

Appendix 3.1 CNC Common Alarm Remedy

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
000	Emergency stop, ESP open circuit	1. Whether the ESP button is controlled	Modify the parameter or check the connection
		2. Incorrect wiring	
		3. The setting of bit 7 of parametr 3003 (ESP) is inconsistent with the actual connection.	
		4. The setting of parameter K10.7 is inconsistent with the actual connection.	
001	Part program open failure	Program is not downloaded before the running in AUTO mode.	Reset to clear alarm and re-execute the programe
002	Part prog. segment loading failure	Error found in loading segment when executing MDI prog. or checking syntax	
010	Single block exceeds 256 characters	Characters excessive in single block	
011	Data exceeds permissive range	Input data exceeds permissive range or the specified data exceeds 8 digits	
012	Address not found	With number or symbol other than address at the beginning of a block	
013	No data follows address	No data follows address or expression format following address checks error, without brackets	
014	Illegal use of negative sign	Sign "-" was input after an address with which it can't be used, or two or more "-" was input	
015	Illegal use of decimal point	Decimal point"." was input after an address with which it can't be use, or two or more "." was input	
016	Input illegal address	Input unusable address in	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		significant area	
017	Incorrect G code	Specify improper G code or that with functions not provided	
018	Address repetition error	Specify the same address twice or more in a block Or specify two or more G codes in same group in a block.Refer to para. 3403#6 AD2.	
019	End of record	Specify end symbol(%) of record, or not specify end of program, refering to para.3404#6 EOR	
040	Too many M codes	Count of M codes specified in a segment exceed value of para.3404#7 M3B or 3	
041	Number followed M code out of range	Digits of M code exceed the value of para.3030 MCB	
042	G code specified error with M99	G28, G30, G53, G36, G37 can't be specified with M99 in a segment	
050	Illegal tool No.	Specify a tool No. which doesn't exist, or exceeds the value of para. 3032 TCB	
051	Compensation No. not found	Tool offset compensation number exceeds range by T code (0~99)	
052	Illegal T code in block	G10、G04 dosn't work with G50(Group A) or G92(Group B)in same block.Refer to param 5006#1 TGC	
060	Feedrate exceeds range	Feedrate was not specified or exceeds range:1.Check G98 or G99 state for feedrate diffrence of usage in Group A 2.Check G94 or G95 state for feedrate diffrence of usage in Group B	
062	Illegal G96 code was found	G96 was specified while const-surface-speed control function is not performed with refference to param.8130#0	
063	Axis specified error in constant surface	In G96 modal, the specified axis by parameter 3770 is wrong	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	speed control		
070	Comand can't run in MDI mode	Command cannot run in MDI mode,for example,G36/G37 、G70~G73	
071	DNC time out	DNC transmission failure, please check	
075	Axes type specified error	Axis type is invalid for specifying; Check the setting of the corresponding parameters 1006#0 and 1006#1 by this axis	
076	Illegal rotaion axis for interpolation instruction	An rotation axis can't be specified except for in polor cordinates、cylinder interpolation、G00、G01 moded	
077	The specified axis is the simple controllable synchronization axis	The specified axis is set to simple synchronization controllable axis in parameter 8311.	
080	Property error for basis axes of plane	Property set error for basis axis of plane in radius interplation	
081	Illegal rotation axes specified in circular interpolation	Modify the program or check the setting of parameters 1006#0 and 1006#1	
082	No radius and I/J/K commanded	In circular interpolation, R, I, J, K has not been specified, refering to para. 3403#5 CIR	
083	Illegal radius	In circular interpolation, Destination is not on the arc specified by R,refering to para.3403#4 RER	
084	Over tolerance of radius	In circular interpolation,difference of the distance between start point and the center of an arc and that between end point and the center of an arc exceeded setting value,refering to para.3410 CRE	
085	Axes too much specified in circular interpolation	In circular interpolation, more than 3 axes specifid	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
086	Three-point arc command data error	There is no instruction in the three-point arc intermediate point instructions, or mid-point instruction can not constitute an arc.	
087	Three-point arc command data can't used to determin full arc.	Three-point arc command can not process full circle, the instruction must be specified end	
088	three-point arc command data error	1.The start、end、mid point shouldn't be on the same line,or start、end point is the sam 2.The radius is 0 dertermined by end、mid point.	
096	Address P or X out of range	Dwell time specified by P exceeds 0~99999999, or X exceeds -9999~9999	
100	Chamfering amount,J was specified error in thread cutting commands	Value of J address exceeds permissive range, or the number followed J is less than zero in G92、G76	
101	Chamfering amount K was specified error in thread cutting commands	Chamfering amount specified by K is less than zero, or exceeds permissive range	
102	Value of L out of range in multi-threading	The value specified in L exceeds the range(1~99) in Multi-threading	
103	Illegal lead command	Lead specified by F is out of range	
104	Value of R out of range in variable threading	In variable threading, the lead incremental and decrementalspecified by R exceeded permissive range	
105	Chamfering amount too large of long axis in threading	Chamfering amount of long axis was greater than thread length; alternatively, the long axis end-retraction value calculated by leading F and parameter 5130 is excessive big in the G92/G76	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		command.	
106	Chamfering amount too large of latitude axis in threading	Chamfering amount of latitude axis in G92 was greater than the distance between start point and end point	
107	Axes not in selected plane in threading	Specify the axis out of the selected panel in thread command.	
108	Illegal axes for interpolation in threading	In threading,basis and paralevel axes are both specified, or more than 2 paralevel axes are specified	
109	C axis not exist in ragid threading	C axis is not set in param No.1020 for ragid threading	
110	C axis is not rotation type in ragid threading	C axis is not rotation type in ragid threading, Refer to param No.1006#0 and 1006#1.	
111	Spindle speed S not specified in ragid threading	S address was not specified for ragid threading command G32.1	
130	Illegal plane select	In the plane selection command, two or more axes in the same direction are selected	
131	Illegal basic axis specified for selected plane		
140	Metric/inch conversion command not at the begining of the program	Metric/inch conversion command must at the begining of the program	
141	Metric/inch conversion command not at a single block	G20/G21 metric/inch conversion can not be shared a same block with other G commands, which should be specified separately.	
142	G20/G21 metric/inch conversion can not be specified in sub-program	The metric/inch conversion is performed during calling the sub-program.	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
150	Improper code in the same block with G22	The G and MSTF code can't be in the same block with G22	
151	For G22 Data exceeds permissive range	Input data exceeds permissive range,\nOr the specified data exceeds 8 digits	
152	For G22 A stroke limit check inhibited area error	The coordinate for para of the plus side inhibited area is not greater than that of the minus side inhibited area Or the difference is not greater than 2000 output increment, refering to para No.1322 & No.1323	
153	G22 command contains an illegal axes instruction	In G22 instruction , axes other than the basic is commanded or U/V/W is used	
154	Axes specified for G22 property error	Instructions the X/Y/Z axis corresponds to the basic property is set to 0 or parallel to the axis	
160	G code in the same block with G25/G26	Specify G code of other group with G25/G26	
165	Reference point not established of axes	Reference point haven't been established before cycle start with referrence to param.1005#0 ZRNx	
166	The axis does not turn to reference point while G28 was specified	The reference point does not set up before performing G28, it is better to modify the program or the parameter 1002#3 AZR.	
167	The axis does not turn to reference point while G30 or G53 was specified	The axis does not turn to reference point while G30 or G53 was specified，Please establish refference point	
168	Illegal reference point by P address in G30	Adderss P specifies other values than 2~4 in G30	
169	Mid point of axis out of range for reference position return in G28/G30	In G28/G30, the position of mid-point is out of range	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
180	Illegal offset value L/P for G10	1.In setting an offset amount by G10, neither L nor P is specified 2. In setting an offset amount by G10, the offset value specified by P is excessive or not specified.	
181	Command address not match in programable param input function	G10/G11 is not match,G10 is specified duplicated,or G11 is specified while G10 is not specified	
182	Proramable param input function not canceled	Proramable param input function is not canceled by G11 before programe ends	
183	Illegal command in Entering data from program	The NC commands, such as the axis address, G code or MSF, etc. are specified in programmed data input.	
200	P value out of range for G31	P value of G31 is beyond 1～4	
201	G31 not allowed in G99	Both basic axes and paralevel axes are specified, or more than 2 paralevel axes for a basic axis are specified	
202	G31 not allowed in tool radius compensation mode	In tool nose radius comensation mode, specify skip cutting command	
210	Illegal G36/G37 specified in auto tool compensation	Illegal G36/G37 specified in auto tool compensation	modify param.6240#7 IGA or modify the programe
211	Offset number not found in G36/G37	Auto tool compensation(G36\G37) was specified without T code	
212	T code not allowed in G36/G377	T code and auto tool compensation(G36, G37) was specified in the same block	
213	Illegal axis command in G36/G37	In auto tool compensation function(G36,G37),an invalid axis is specified	
214	Illegal axis command in G36/G37	Axis specified to move is not the corresponding axis in G36、G37 or the command is incremental	
215	Param error in	γ is less than ε.Refer to param.	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	G36/G37	6251 ATOR1, 6254 ATOE1, 6252 ATOR2, 6255 ATOE2	
216	ATC not allowed in tool radius compensation mode in G36/G37	Auto tool compensation(G36、G37) was specified in tool radius compensation mode	
230	Illegal command in the same block with G7.1	In circular interpolation G7.1（G107）block，other group of G codes or MST is specified	
231	None-rotaion axis specified to start circular interpolation	None-rotation axis is specified whith G7.1(G107) block	
232	Too many rotation-axes specified in circular interpolation	Too many rotation-axes is specified with G7.1(G107) in block	
233	Illegal negative sign of radius specified in circular interpolation	Illegal negative sign of radius is specified with G7.1(G107) in block	
234	Illegal G12.1, G51.2 found in circular interpolation	In circular interpolation mode,it is illegal to specified polar interpolation command G12.1,or polygon processing command G51.2	
235	Illegal change-plane command found in circular interpolation	In circular interpolation, it is illegal to specified G17~G19 to change plane	
236	Illegal change-workpiece-cordinates command in circular interpolation	In circular interpolation,it is illegal to specify G54~G59 to change workpiece cordinates	
237	Illegal multi-cycle command specified in circular interpolation	In the column interpolation method, G54~G59 can not be specified to perform the workpiece coordinate system	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		selection	
238	Illegal tapping, drilling command specified in circular interpolation	Illegal tapping、drilling command is specified in circular interpolation by G84~G89	
239	Illegal canned-cycle command specified in circular interpolation	Illegal canned-cycle command specified in circular interpolation G90~G94	
240	Illegal threading command specified in circular interpolation	Illegal threading command specified in circular interpolation by G32 or G34	
241	Illegal axis of other plane specified in circular interpolation	Illegal axis of other plane is specified in circular interpolation	
242	In circular interpolation mode,G code in Group 00 other than G04 is specified	In circular interpolation mode,G code in Group 00 other than G04 is can't be specified,including G27~G30,G31,G36/G37 ,G52 ,G 53	
243	Ilegal G code or T code specified in circular interpolation	In circular interpolation,it is illegal to specify G00, or T code	
244	Specify radius by I/J/K in cylinder circular interpolation	Specify radius by I/J/K in cylinder circular interpolation	
245	Improper code in cylinder circular interpolation in tool compensation C type	When in G41 or G42:specify improper G-code to start or end circular interpolation	
250	Param error for polar interpolation	When the polar coordinate interpolation command G12.1 (G112) is performed, the setting of the corresponding parameter 5460 (linear axis) or parameter 5461 (revolving axis) by polar	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		coordinate interpolation axis is detected.	
251	C-type tool compensation error in polar interpolation	When C-type compensation is performed(None-G40-Modal),it is illegal to specify polar interpolation by G12.1/G13.1	
252	Illegal axes specified for selected plane in polar interpolation	In the polar coordinate interpolation method, the axis of the arc command is out of the selected panel.	
253	Other group of G code in sanme the block with G12.1/G13.1	Specify the G command of other groups in the polar coordinate interpolation command G12.1/G13.1 (G112/G113) block.	
254	Repetition of G12.1 command.	G112 already performed while command another G112	
255	Illegal T instruction found for polar interpolation	Specify T code which can't be used in polar interpolation.	
256	Illegal G code specified in polar interpolation	In polar interpolation, only G codebelow is proper: 1.Group 00:G04 or G65. 2.Group 01:G01、G02 or G03. 3. Group 03(G98/G99 in A-type-gcode, G94/G95 in B-type-gcode). 4. Group 05:G40~G43, and Group 09:G66/G67.	
270	G50 is invalid	Alarm when specify coordinate set command(G50 in A-type-Gcode,G92 in B-type-Gcode),refer to param 1202#2	
271	Value to set the coordinate system is out of range	Refer to the valid range of G50(G92 in B-type-Gcode)/G52/G53/G54~G59	
272	Address P out of range for Append cordinate	Address P to select append cordinate is out of range in G54.1	
280	Address P not defined	Address P(program number) was not commanded in block	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		inlcuding M98, G65 or G66	
281	Subprogram nesting error	The subprogram call exceeds 12 folds	
282	Program number or sequence number not found	The program number was not found specified by P in M98, M99, G65 or G66.	
283	Fail to open the program during sub-program calling	Sub-program calling; fail to open the sub-program when the internal preread sub-program occurs.	
284	Fail to init nc buffer of sub program	Fail to buffer the initial sub-program when sub-program calls.	
285	Subprogram call error	A program can't call main program or itself in M98,G65 or G66	
286	Subprogram in use	Sub program can't be called because it is in edit state or unsave state	
287	Program call statement can't run in MDI&DNC operation	Marco program and subprogram call in MDI &DNC operation isn't supported	
300	Illegal G code specified with multi-spindle control	When select spindle in multi-spindle control function, illegal G code is specified with S_P_;S_ and G25/G26、S_P_ and G96/G97 can't be specified in same block	
301	P address error in multi-spindle selection	When select spindle in multi-spindle control function, P address assigns an illegal value beyond the range set in Param No.3781	
302	Absence of P address in multi-spindle selection	In multi-spindle selection,P address with spindle speed command S is absence.The alarm is release when Param No.3706#2 is set to 0	
303	Error multi-spindle function disabled	1.Param MPP（No.3703#3）=0,while specify P address to select spindle\n2.Param	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		MPS(No.3781)=0，while specify P address to select spiondle	
310	Illegal polygon processing command in C-type tool compensation mode	In C-type tool compensation mode(None-G40 mode),while spcify polygon processing commandG51.2(251)	
311	G51.2 repetition in polygon processing	In the G51.2 polygon machine mode, the command, G51.2, is specified again.	
312	Error found in P address of G51.2	P is not spcified or set by a value out of range in G51.2	
313	Error found in Q address of G51.2	Q is not spcified or set by a value out of range in G51.2	
314	PQ ratio incorrect	P/Q is not equal to 0 refer to param7603#1.	
315	G51.2 is not perfomed in polygon processing	When Param No.7610 is set to 0,polygon processing command G51.2 is not used	
316	Rotation axis set error	The shaft axis of rotation is not set	
317	Conflict code specified in the same block with G51.2 or G50.2	G51.2 or G50.2 is specified in the same block with other G or MT instructions	
318	polygon processing instructions of the screw mosquito instructions	Polygon processing, command a mosquito Lo instructions	
319	polygon processing command axis command is illegal	Polygon processing, command the tool rotation axis move command	
320	polygon rotation axis machining work is no instruction S instruction position mode	Polygon rotation axis of the workpiece machining way to position mode (para 7603 #4), there is no instruction S command.	
321	Illegal command specified in polygon processing	In polygon processing,circular interpolation command G7.1, polar interpolation command G12.1 and tapping/drilling cycle	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		command G84~G89 can't be used	
400	Parameter swtich is ON	Press【RESET】to cancel alarm.	
401	Fail to open the component program	The current program is removed or there is no current program.	
402	Parameters back up failure	Check the memory or power-on again	
403	Parameters recover failure.	Check whether parameters are being written in, or power-on and retry	
405	Recovery of system param successfully	Please power on again	
406	Recovery of PLC param successfully	Please power on again	
407	Recovery of servo param successfully	Please power on again	
408	Recovery of io units param successfully	Please power on again	
409	Fail to import param from extern file	Data in imported param file detected invalid, and old data was recovered	
430	More than 3 parameters fount out of range	More than 3 parameter data are exceeded the setting range, which are already used the default value.	
450	Parameter is already modified.	.	A parameter which requires the power off was input, turn off power
451	Servo parameter is already modified.		A servo parameter was modified which requires the system & the servo restart
452	Bus communication logic ID number is modified.		Before the device has been restarted,the internal station address will not be effective
453	I/O unit parameter is already modified.		I/O parameter which requires the power off was input, turn off power
460	Number of CNC controllable axes		Check para No.1010 and 8130

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	exceeds the total number		
461	Duplicated axis attribution were set		Modify pra No. 1022
462	Duplicated axsi name were set	The possible reasons: 1) The same axes names are set; 2) The forbidden axis name is specified at the current G code system.	Modify the parameter No.1020 or No.3401#6.
463	Disabled rotation axis setting	The setting of rotation axis of parameter No.1006 is disabled	
465	Duplicated servo comm id was set for control-axis		Modify para.No.1023
470	Duplicated servo comm id or analog address was set for spindle		Modify para.No.3717
471	Invalid AO address of io uint was set for spindle		Modify para.No.3717
472	Duplicated spindle encoder number was set		Modify para.No.3723
474	Logic id was not set for spindle using Cs contour function		Modify para.No.3717 or 8133#2, 3704#6, 3704#7.
475	,Logic id was not the same between axis and spindle using Cs contour function		Modify para.No.1023、3717, 8133#2, 3704#6, 3704#7
476	Logic id was the same between axis and spindle while Cs contour function is not used		
477	Ao address of spindle beyond the maxium ao address of the correspong I/O unit		Modify para.No.3717.
480	Duplicated comm id		Modify para.No.3051,

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	of io unit was set for spindle		No.3052, No.3053, No.3054
490	None of valid comm It was set		Modify para.No.9000#0, or No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, or No.3060
491	Custom macro config file changed, restart the system to become effective		The custom macro config file has changed, restart the system to make it effective
500	Reference position not established		Manually move to reference return and press "axis move" key on operation panel under "referrence return" mode to establish it
501	Encode data error		It is necessary to set up the machine reference point; it may be the reason of the encoder data read error.
504	Servo battery voltage too low		Please replace the servo battery , and then reestablished the refference position
510	The alarm occurs due to the extensive error on the power-on machine coordinate	The error value between the machine tool coordinate establishement and memory with power-on exceeds the tolerance. Reason: 1) The carriager position moves during the machine tool is power-off; 2) The setting value of parameter No.1206 is excessive small.	It is necessary to reset the machine tool reference point or turn the power on again.
511	Machine cordinate initialized error too large	The error value between the machine coordinate establishment and memory with power-on is excessive big. Reason: 1) The carriager position moves during the power-off, turn on the power again. 2) The motor	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		encoder is changed.	
512	The system parameter relevant to the reference point has been altered.	The numerical value is different when the system parameter and the machine reference point establishment are inspected with power-on, which includes parameter No.1811#2 or No.1816 or No.1820	
513	The servo parameter relevant to the reference point has been altered.	The numerical value is different when the servo and the machine reference point establishment are inspected with power-on, which contains of the command reverse parameter or gear ratio parameter.	
514	Fail to read the parameter record file relevant to the reference point	The parameter value can not be checked the previous established machine zero when the power is turned on.	
520	PC signal detected error because of servo dislink		Please check communication connection of gsklink, and power on again
604	Servo alarm		Check the servo
650	Power supply to the servo is turned off	The coordinate system became inacurrate when the control command to the servo is interrupted	Please return to the reference postion
700	stored stroke limit1 : +	Exceeded the + sides stored stroke limit 1	
701	stored stroke limit1 stored stroke limit1 : -	stored stroke limit1	
702	stored stroke limit2 : +	Exceeded the + sides stored stroke limit; 2. The internal/external inspection is determined by bit 0 of parameter No.1300.	
703	stored stroke limit2 : -	Exceeded the - sides stored stroke limit; 2. The	

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		internal/external inspection is determined by bit 0 of parameter No.1300	
704	stored stroke limit3 : +	Exceeded the + sides stored stroke limit 3	
705	stored stroke limit3 : -	Exceeded the - sides stored stroke limit 3	
706	Over travel : +	Exceeds + side overtravel limit	
707	Over travel: -	Exceeds - side overtravel limit	
710	Spindle speed alteration inspection alarm	Actual spindle speed exceeds the allowable range of commanded value	Check the machine tool cutting state, or refer to param No.4912 and 4913
720	Error in manual tool offset measurement	Illegal operation including: 1. A couple or more of axes shift has beed detected, or no axis shift has been detected while the complete signal inputed; 2. Derection of axis shift and complete signal detected reversal; 3. Derection of axis shift was not fixed; 4. Another signal inputted while the last measurement was not completed; 5. At present, the manual tool-setting operation is being operated	
721	Referrence position not established for manual tool measurement	Referrence position of the axis is not established	Establish the REF. position first and then re-measure
722	Data written error in manual tool measurement	Data write error in the manual tool measurement	Please check the ram
723	Error selecting tool offset number by PLC in manual tool measurement	Tool offset number selection is incorrect by PLC in the manual tool measurement	Please check the PLC
905	Append cordinate	G54.1 append cordinate failed to	Press 【RESET】to cancle

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	established failure	established	alarm, or power-on again or send it to the factory for inspecting.
910	Initial parameter failure	User parameter file does not exist or data is damaged.Default parameters become effective	
911	Initial CNC configuration failure	CNC config file does not exist or data is damaged. Default configuration becomes effective	
912	Initial tool offset data failure	Tool offset file does not exist or data is damaged. Initial data becomes effective	
913	Initial tool life data failure	Tool life file does not exist or data is damaged. Initial data becomes effetive	
914	Initial pitch error compensation data failure	Pitch error compensation file does not exist or data is damaged. Initial data becomes effective	
915	Initial PLC programe failure	Read file failure in registering program, or compile failure.	
930	Tool compensation initialization failure	The tool compensation file is not available or the verifed error.The backup value is loaded	
931	Parameter file initialization failure	The Parameter file is not available or the verifed error.The backup value of parameters is loaded	
940	Data in NVRAM changed	Data version in nvram detected inconsistent with the new version in used	Data version in nvram detected inconsistent with the new version in used.\nFor ABS. encoder is used, please re-establish the REF. position; For INC. encoder, please return to REF. position again. Please re-excute the PLC progame and using the default data in plc to recover the data in NVRAM

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
941	Power off when accessing NVRAM	The system powered off as running	Perform REF. position return, and check the coordinates, tool offset values
942	Data area 1 of NVRAM detected abnormal	Verification of data area 1 of NVRAM is wrong or ruined	Perform REF. position return, and check the cordinates 、 tool offset values. If this alarm frequently occurs, it is better to maintain it by factory.
943	Data area 2 of NVRAM detected abnormal	Verification of data area 2 of NVRAM is wrong or ruined	If the system uses the absolute encoder, rebuild the reference point. If the system uses the incremental encoder, the zero return operation should be operated again. If the alarm frequently occurs, it is better to maintain by sending to the factory.
950	TRYOUT limit timed out. System functions are restricted.		Please contact the dealer
990	Too many alarm and info	The number of alarm exceeds 20 or number of info exceeds 30.	
991	Undefined alarm No.	Missing alarm content for alarm No.	
992	Format error in alarm content	Part of data in alarm content and operation info was incorrect	
993	PLC alarm information table error	The alarm num specified is not found in PLC alarm information table or out of the range 1000~2999	Modify the PLC alarm information table
4000	Syntax check cancelld because of reset	Reset causes incompleted the syntax check, refering to para 3401#2 NCK	Please check again
4001	Same sequence number found in syntax check	There are duplicated sequence number, which might cause error	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
4010	Value of some paramter out of range	This may be caused by system updating, or switching of param ISB\ISC, or switch of linear axis and rotation axis	
4011	GSKLink function is not used,while valid station address was set		Modify para No.9000#0,or No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, or No.3060
4012	Custom macro config data absence	The custom macro displaying function is used,while the config file is not found, or no config data in it	Modify the Param No.8132#6, or execute the config file custom macro window
4013	Custom macro config data error	The custom macro displaying function is used,while error configuration found in the config file,Please execute the correct config file	
4020	Default encoder communicates through GSKLink while none of valid communication station address has been set for the spindle		Modify para.No.3717, or set a none-zero value to para.3723
4100	Fail to set FPGA param	The FPGA param was not set successfully which may cause abnomal in threading or spindle control	Please power on again or contact the manufacturer
4110	Servo battery voltage low	The position will lost when power off the servo	Troubleshooting: Change the battery. Note: It is important to change the battery in the state of (that is, the servo driver is power-on) the driver alarm displays. And then turn it off, the alarm eliminates after the power is turnd on. Explanation: User can continue machining after

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
			this alarm occurs, but it is essential to change the battery as soon as possible.
4120	Gateway GSK-Link-PA terminal slave has been altered.		The gateway and its corresponded communication link need to be restart。
4200	Machine panel devID error		Check the connection between machine panel and CNC
4201	Machine panel device infomatin error		Check the connection between machine panel and CNC.
4202	Machine panel continous communication error detected		Check the connection between machine panel and CNC
4205	Soft panel enabled,and machine panel stoped		Refer parameter No.8136#3(NOP) to enable machine panel
4210	MDI panel communication error	MDI panel communicated with CNC failure	
4300	System enters level 1	Never attempt to modify the parameter when the system is the level one authority.	
4304	Default password for try-out funtion detected	Please change the password for try-out function for security	
4305	The system will reach the try-out time limit	The system will stop soon.Please contact the salsman to get release code	
5000	Communication disconnect physically	Disconnection or interfrence on calble may cause the alarm	Power on again
5001	Check of ring devices overtime	Disconnection or interfrence on calble may cause the alarm	The system has re-tried to fix the problem.Even so,it may fail at last.Please refer to final state of communication to [RESET] the alarm, or power on

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
			again
5002	Handshake of ring B failure	Disconnection or interfrence on calble may cause the alarm	The system has re-tried to fix the problem.Even so,it may fail at last.Please refer to final state of communication to [RESET] the alarm, or power on again
5003	Check of time-delay failure	Disconnection or interfrence on calble may cause the alarm.	The system has re-tried to fix the problem.Even so,it may fail at last.Please refer to final state of communication to [RESET] the alarm, or power on again
5004	Communication configuration param error	Disconnection or interfrence on calble may cause the alarm.	The system has re-tried to fix the problem.Even so,it may fail at last.Please refer to final state of communication to [RESET] the alarm, or power on again
5005	GSKLink initial error		Please power on again
5006	Devices number detected unequal to value set in system params		Refer to PAR.No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, No.3060, and comm station address of each device. Power on again.
5007	None of valid comm station address of device detected		Refer to PAR.No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, No.3060, and comm station address of each device. Power on again
5008	Communication returned to CP0 on master station		Power on again
5010	Incorrect parameter for GSKLink		Please refer to PAR.No.1023, No.3717,

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
			No.3050, No.3051, No.3052, No.3053, No.3054, No.3060, and comm station address of each device.Power on again
5011	GSKLink disconnected		Please check the connection to each device.Power on again
5020	MDT lost		Please check the device
5021	MST lost		Please check the device
5022	MDT data verified error		Please check the device
5023	GDT data verified error		Please check the device
5030	C1D device alarm		Please check the device
5031	C2D device alarm		Please check the device
5040	Communication of slave device stopped		Please check the device
5100	IDN16,24 error		Refer to coresponding system parameters, and check the work state of the device
5101	IDN32,35 error		Refer to coresponding system parameters, and check the work state of the device
5102	IDN5030,5031,5033 error		Refer to coresponding system parameters, and check the work state of the device
5103	Fail to config I/O unit		Please check the devic.
5132	C3D device alarm	This may be caused by：(1)Modify servo param on servo side;(2)Re-load\recover servo params on servo side;(3)Save servo params on servo side	Press [RESET] to release the alarm
5133	Please check the device.Press		

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	[RESET] to release the alarm		
5198	Initializing GSKLink	Please wait	
5199	GSKLink communication error.		
5200	Fail to load servo property		Power on again.
5201	Fail to load servo infomation lists		Power on again
5210	Fail to load servo parameters		Power on again
5211	Import of servo parameters	Please select the import param to be effective by【SELCT EFF. PAR】 softkey. The parameters could not be saved before selecting.	
5220	Inconsistent of servo param read from servo-param-file saved in cnc and that loaded from servo		Please enter [servo param] layer and then select the effective servo param with【SELCT EFF. PAR】softkey.
5400	IO unit param file not exist	IO unit param file not exist, And the file failed to be created automatically	Press [RESET] to release the alarm
5401	Mapping table does not record the correct read configuration	It is already automatically written the current read configuration.	Press [RESET] to release the alarm
5402	The mapping table detects the incorrect address or parameter setting		The error has been corrected automatically Press [RESET] to release the alarm.
5403	Fail to load property	Fail to read the information from remote equipment, check whether the I/O unit is on the	Power on again

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		normal working state.	
5404	Inconsistent	The property of I/O unit loaded is different from before.Current property is recorded automatically	Press [RESET] to release the alarm.
5406	Fail to config the device	Fail to send the parameter to the remote equipment, check whether the I/O is on the normal working state.	Power on again
5500	Fail to load property of gateway		Power on again
6000	Data exceeds permissive range of extern coordinate origion offset	Data exceeds permissive range	Modify the programe
6001	Data exceeds permissive range of tool offset	Data exceeds permissive range	Modify the programe
6005	Data exceeds permissive range of additional workpiece cordinate origion	Data exceeds permissive range	Modify the programe.
6006	Error loading Append cordinate offset	Fail to read the additional workpiece coordinate system offset value when the workpiece coordinate system is updated.	Please check the cnc flash
6007	Data exceeds permissive range of workpiece cordinate origion	Data exceeds permissive range	Modify the programe.
6010	Toollife data runed error	Tool Group No. exceeds the maxinmum allowable value,or the tool group commanded in the machine program is not set	Modify the program or modify the tool life data
6015	Auto tool compensation signal Not detected	In auto tool compensation mode(G36、G37),when enter the area assign in param,The measurement arrival signal is not detected (XAE\EAE)	Refer to settings or operation

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6020	Over-speed of spindle in threading	In threading, the spindle speed specified is too fast for the threading axis	Modify the program
6021	Spindle speed too low in threading	S command was not specified or is set to zero, Spindle encoder feedback is abnormal	Modify the program
6022	Signal of 1-rotation not detected	The specified speed by spindle is lower when the thread machining is performed, which causes the feed axis abnormal.	Modify the program
6023	Increase/decrease amount of lead error in threading	Increase/decrease amount of lead is too large,which causes the feed axis abnormal	Modify the program
6024	Spindle encoder lines out of 100~5000	This type of encoder is not supported in taping	Check the parameter setting (NO.3720 and NO.3723) or change the spindle encoder
6025	Spindle rotation signal(SFR,SRV) detected error in taping	Check the output of G signal SFR, SRV or encoder connection.	Modify the program or PLC
6026	The spindle speed is excessive high or low during the common tapping, so that the tapping axis can not be performed the feed normally.	The possible reasons: 1) S command specified a value equeal to zero or out of range; \n2)Encoder feedback abnormal 3) Abnormal spindle encoder feedback.	Modify the program or check the status of encoder
6027	One-revolution signal Not detected in tapping	Refer to pram NO.3723 for correct encoder setting of spindle	Check the work-state or connection of the encoder
6028	M code execution abnormal for spindle start	stop in tapping\nRefer to PLC for M code processing with spindle start/stop,CW/CCW rotation	Check whether the M code has become effective
6029	Incorrect spindle encoder selection	Incorrect spindle encoder selection, so that the thread or tapping can not be performed.	Modify the program or check the parameter NO.3723.
6030	Raggid tapping signal is off	The probable reasons: 1) Fail to detect the RGTAP signal or do not specify (By the M29 or other M codes) the rigid tapping mode	Modify the program or check the ladder diagram

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		before the tapping is performed; 2) Fail to correctly send the rigid tapping spindle selection signal.	
6031	C-axis commanded in spindle mode when execute ragid tapping	The program specified a movement along the Cs-axis when the signal CON (G27#7) is OFF.	Correct the program, or consult PLC program to find the reason the signal is not turned on
6032	Spindle selection error in ragid tapping	Refer to following possible caution: 1) Check the Plc for correct spindle selection signal assignment. 2) Refer to Param for correct Cs contour setting	
6035	C-axis commanded in spindle mode	The program specified a movement along the Cs-axis when the signal CON (G27#7) is OFF.	Correct the program, or consult PLC program to find the reason the signal is not turned on
6036	C-axis commanded in spindle mode when execute G28	The program specified a movement along the Cs-axis when the signal CON (G27#7) is OFF.	Correct the program, or consult PLC program to find the reason the signal is not turned on
6037	Spinle speed arrival signal Not detected	Spindle speed arrival signal(SAR) is not detected in cutting	Modify the programe or PLC
6038	Error detecting the release signal for ragid tapping (RGTAP)	RGTAP signal error in cancel ragid tap	Refer to PLC or Param No.5200#2
6040	spindle	In polygon processing, the rotation axis for workpiece spindle is not set, or Cs contour setting is incorrect, or the rotation axis for workpiece conflicts with the axis for tool rotation.	Modify the Plc for spindle selection, or Refer to Param No.7610
6041	synchronous instruction is illegal in polygon processing	In the polygon processing, synchronous operation, the synchronous axis movement command is issued by the NC program	please modify the program
6042	synchronous mode command is illegal in polygon	Polygon processing, while trying to synchronize the operation and CS contour contro	please modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	processing		
6043	Not in position control of workpiece rotation axis in polygon processing	The workpiece rotation axis doesn't enter position control mode in polygon processing	Modify the programe or Check the PLC for the reason why the signal doesn't become ON
6044	olygonal cutting spindle speed error	In the polygon processing method can not maintain the rotational speed ratio command value, since the spindle speed or faster than the polygonal shaft clamp synchronous or low	please modify the program
6045	Not in position control of tool rotation axis in polygon processing	The tool rotation axis doesn't enter position control mode in polygon processing.	Modify the programe or Check the PLC for the reason why the signal doesn't become ON
6050	Illegal variable number in macro program	A value not defined as a variable number is designated in the custom macro	Modify the program
6051	Macro varables are protected	The macro varable is protected from modification	Refer to PARA. 6031 and 6032
6052	Macro varables modification is forbidden	The macro value is read-only	Modify the programe
6053	Null value not allowed for system macro	Null value is not allowed to set to system macro	Modify the programe
6054	Data exceeds permissive range of macro value	Data exceeds permissive range	Modify the data
6060	The same axis was commanded by PLC and CNC	Axis control command was given by PLC to an axis controlled by CNC	Modify the program or check the PLC.
6061	Cannot change PLC control mode	Select an axis which is in commanding by PLC control	Modify the PLC program
6070	Encoder data error becaused of PC signal Not detected	The servo is not applied,or Gsklink communication failed,or the spinle failed to rotate to the precise postion for correct encoder data	Check the Gskling communication state or the work state of spindle
6075	Error return to reffrence point of slant-controled axis	In the state of the manual reference point return in the slopping axis control and no	Perform the reference point return operation of the perpendicular axis after its

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		reference point return operation after power-on for the automatic reference point return, try to perform reference point return operation of the perpendicular axis in the state of the slopping axis reference point return does not execute.	operation along the slopping axis is completed.
6080	Illegal G code specified in handwheel retraction block	Illegal G code specified in handwheel retraction block was found	Modify the programe
6200	Canned cycle cmd in non ZX plane	Canned cycle can't commande in non ZX plane	Modify the programe
6201	Specify other axes not inlcuded in ZX plane	Specify other axes not included in ZX plane	Modify the programe
6202	In G90/G77,G92/G78 commands, absulute value of R is greater than that of address U(radius assigned)	In block using G90/G92 command （ Gcode Group B:G77/G78 ） ,When sign of address R and U is opposite,absolute value of R is greater than that of U(Radius assigned)	Modify the programe
6203	In G94/G79 commands, absolute value of R is greater than that of W.	In blocks using G94（Gcode Group B:G79）,When the sign of address R and W is opposite,the absolute value of R is greater than that of W	Modify the programe
6210	Illegal plane select in multple repetive cycle	When specifying the multi-cycle G70~G76, its plane does not XZ (modal regards as G18), or specify the plane shifting commands G17~G18 in G70~G76 blocks.	Modify the programe
6211	Specify other axes not inlcuded in ZX plane in G70~G76	Specify other axes not inlcuded in ZX plane in G70~G76	Modify the programe.
6212	Illegal G code in G70~G73	In the G70~G73, unused G code is specified between two blocks based upon the addresses P and Q; the G0~G3, G96/G97,	Modify the programe

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		G98/G99, G40~G42can only be specified in the blocks of NS~NF.	
6213	G70~G73 cannot operate in MDI mode	G70~G73 with P & Q was specified in MDI mode	Modify the programe
6214	Illegal macro statement in G70~G73	Macro statement is unallowable in G70~G73 command	Modify the programe
6215	Call the sub-program in the G70~G73 cycle	Fail to call the sub-program in the G70~G73 cycle	Modify the programe
6216	Illegal subprogram call in G70~G73	Subproram call is unallowable in G70~G73 command	Modify the programe
6217	Incorrect address P command in G70~G73	Fail to specify the P or the command exceeds its range in the G70~G73.	Modify the programe
6218	Incorrect address Q command in G70~G73	Fail to specify the Q or the command exceeds its range in the G70~G73.	Modify the programe
6219	Fail to search the address P or Q in the G70~G73	The sequence number specified by P & Q was not found in G70,G71,G72 or G73	Modify the programe
6220	The commands between P and Q are same in the G70~G73	The number specified by address P & Q the same in G70~G73	Modify the programe
6221	The two blocks as components of G71~G73 command are discontinuous	The two blocks as components of G71~G73 command are discontinuous, which is possible to cause error	Modify the programe
6222	Blocks between Ns & Nf exceeds 100 in G70~G73	Too many blocks of the Ns—Nf in G70~G73, it exceeds the Max. allowable 100 blocks.	Modify the programe
6223	Cutting direction determined by Ns-Nf blocks is the same with track direction in	Direction of cutting conflicts with track direction,so that the track will not close	Modify the programe

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	G71~G73		
6224	Direction of cutting(in Ns-Nf blocks) and finishing allowance is the same in G71~G73	Direction of cutting(in Ns-Nf blocks) and finishing allowance is the same in G71~G73	Modify the programe
6225	Shape specified in Ns-Nf blocks Not monotonous in G71~G73	Arc specified in Ns-Nf blocks of G70~G73 is long arc	Modify the programe
6226	Arc shape specified in Ns-Nf blocks Not monotonous in direction of X axis in G71~G72	Arc shape specified in Ns-Nf blocks is not monotonous in direction of X axis in G71type-I or G72	Modify the programe
6227	Arc shape specified in Ns-Nf blocks Not monotonous in direction of Z axis in G71~G72	Arc shape specified in Ns-Nf blocks is not monotonous in direction of Z axis in G71type-I or G72	Modify the programe
6228	Arc shape specified in Ns-Nf blocks Not monotonous in direction of X axis in G71 type-II	Arc shape specified in Ns-Nf blocks is not monotonous in direction of X axis in G71 type-II	Modify the programe
6229	Arc shape specified in Ns-Nf blocks Not monotonous in direction of X axis in G73	Arc shape specified in Ns-Nf blocks is not monotonous in direction of X axis in G73	Modify the programe
6230	Arc shape with finishing allowance specified in Ns-Nf blocks Not monotonous in direction of Z xis in G73	Arc shape with finishing allowance specified in Ns-Nf blocks is not monotonous in direction of Z xis in G73	Modify the programe
6231	Arc shape with retraction amount specified in Ns-Nf blocks Not	Arc shape with retraction amount specified in Ns-Nf blocks is not monotonous in direction of Z xis in G73	Modify the programe

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	monotonous in direction of Z xis in G73		
6233	X axis of start point was on cutting path in G71/G72	Start point was on cutting path in G71/G72, which may cause interfere of tool and workpiece, refering to para. 5104#2 FCK	Modify the programe
6234	Z axis of start point was on cutting path in G71/G72	Start point was on cutting path in G71/G72, which may cause interfere of tool and workpiece, refering to para. 5104#2 FCK	Modify the programe
6235	X axis of start point was on cutting path in G73	Start point was on cutting path in G73, which may cause interfere of tool and workpiece, refering to para. 5104#2 FCK	Modify the programe.
6236	Z axis of start point was on cutting path in G73	Start point was on cutting path in G73, which may cause interfere of tool and workpiece, refering to para. 5104#2 FCK	Modify the programe
6237	Too many concaves in G71 type II	More than 10 concaves are specified in G71 type II	Modify the programe
6238	Direction of chamfering and finishing allowance along X axis is inconsistent in G73	Direction of chamfering and finishing allowance along X axis is inconsistent in G73	Modify the programe
6239	Direction of chamfering and finishing allowance along Z axis is inconsistent in G73	Direction of chamfering and finishing allowance along Z axis is inconsistent in G73	Modify the programe
6240	Too many cutting blocks per cycle in G71 II cycle	Too many cutting blocks per cycle are specified in G71 II cycle	Modify the programe
6241	Finishing allowance U of X axis in G70~G73 beyond proper range	Finishing allowance U of X axis in G70~G73 is beyond proper range	Modify the programe
6242	Finishing allowance W of Z axis in G70~G73 beyond proper range	Finishing allowance W of Z axis in G70~G73 is beyond proper range	Modify the programe

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6243	G00 or G01 move command not found in first block of G71~G72	G00 or G01 move command should include in first block of G71~G72	Modify the programe
6244	G00-G03 move command not found in first block of G73	G00-G03 move command not found in first block of G73	Modify the programe
6245	Over tolerance of radius in G71~G73	The radius D-value calculated by arc command in G71~G73 exceeds its range.	Modify the programe
6246	X axis motion in the first block of G71	X axis increment was not commanded in first block of G71, or X axisincrement is zero	Modify the programe
6247	Z axis motion in the first block of G72	Z axis increment was not commanded in first block of G72, or Z axis increment is zero	Modify the programe.
6248	Depth of cutting is less than zero or more than maximium in G71 or G72	Single tool infeed value is less than or equals to 0 in G71 or G72 command, alternatively, it is more than the top tool in-feed value.	Modify the programe
6249	Escaping amount(R(e)) is less than zero in G71 or G72	Escaping amount(R(e)) is less than zero in G71 or G72	Modify the programe
6250	Increment cutting amount out of range in G73	Increment cutting amount out of range in G73	Modify the programe
6251	The number of division R(d) in G73 out of range	The rounding number of division is less than 1 or more than 999	Modify the programe
6252	Direction of cutting and finishing allowance is the same	Cutting direction determined by position point and NS block conflicts with the finishing allowance direction	Modify the programe
6253	Para. modified failure in G71~G76	Para. modified failure in G71~G76. Check that the para. file be abnormal	Modify the programe
6254	Part prog. segment loading failure in G71~G76	Fail to read the program in NS~NF from G70~G76 during operation	Modify the programe
6260	Value of address Q	Value of address Q is beyongd	Modify the programe

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	beyongd the proper range in G74G75	the proper range in G74G75	
6261	Value of address P beyongd the proper range in G74G75	Value of address P is beyongd the proper range in G74G75	Modify the programe.
6262	Value of address R(e) is beyond the proper range in G74 or G75	Retraction amount specified by address R(e) in G74 or G75 is less than zero, or greater than maxmium.	Modify the programe.
6263	Value of address R(△d) is beyond the proper range in G74 or G75	Retraction at the cutting end specified by R(△d) in G74 or G75 is less than zero, or greater than maxmium	Modify the programe
6270	X or Z axis increment is 0 in G76	X or Z axis increment is 0 in G76	Modify the programe
6271	Repetitive count in finishing is less than 1 or greater than 99 in G76	Repetitive count in finishing is less than 1 or greater than 99 in G76	Modify the programe
6272	G76 thread chamfering width P(r) exceeds the permit range.	G76 thread chamfering width P(r) exceeds the permit range.	Modify the programe
6273	Angle of tool tip out of range in G76	Angle of tool tip out of range in G76	Modify the programe
6274	Q(△dmin) out of range in G76	Minimum cutting depth Q (△dmin) out of range in G76.	Modify the programe
6275	Finishing allowance R(d) out of range in G76	Finishing allowance R(d) is less than least increment in G76	Modify the programe
6276	G76 thread taper R(i) beyongd the proper range	G76 thread taper value specified by address R(i) exceeds the proper range	Modify the programe
6277	R and U is inconsistent for taper thread cutting in G76	Machining start position is between thread beginning point and end point in G76	Modify the programe.
6278	Thread height not specified by P in G76	Thread height not specified by P in G76	Modify the programe.

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6279	Incorrect thread height in G76	Thread height is less than Finishing allowance in G76	Modify the programe
6280	Thread height is less than Finishing allowance or minimum cutting depth in G76	Thread height is less than Finishing allowance or minimum cutting depth in G76	Modify the programe
6281	Thread height is larger than the destination to end point in G76	The tooth specified by G76 is more than the distance between the positioning point and thread end.	Modify the programe
6282	Number followed address Q is out of range in G76	epth of cut in 1st cut Q was out of range, or not specified	Modify the programe
6283	Taper of thread is biger than 45 in G76	Taper of thread is biger than 45 in G76	Modify the programe
6284	Taper is paralevel to the tool in G76	The thread tapper specified by G76 is parallel with the cutter, and it can not be performed the cutting.	Modify the programe
6285	Incorrect thread taper or pointed angle specified by G76	G76 can not be specified the correct pointed angle or thread tapper, so that the normal cutting can not be performed.	Modify the programe
6300	Illegal S code command in rigid tapping	In rigid tapping,an S value is out of range or not specified	Modify the programe
6301	S code not found in rigid tapping by G84 or G88	In G84 or G88 rigid tapping(parameter 5200#0 is set to 1),an S value is not specified	Modify the programe
6302	Beyond the range of address J for spindle extraction in ragid tapping	Value of address J exceeds the range for spindle extraction in ragid tapping	Modify the programe
6303	Illegal K in tapping	The specified repeated times K value in the tapping or drilling canned cycle does not within the 1~99.	Modify the programe
6304	Lead specified in address F beyond the range in tapping	Lead or speed specified in address F beyond the range in tapping	Modify the programe

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
6305	Inch Lead value specified in address I beyond the range in tapping	Inch Lead value specified in address I is beyond the range in tapping	Modify the programe
6306	Incorrect program command in the rigid taaping	In the rigid tapping, the M code and S value based upon the rigid tapping mode does not share with a same block.	Modify the programe
6307	Illegal axes-motion command in ragid tapping mode	In ragid tapping, a motion block is specified between M code(start ragid tapping) and G84 command	Modify the programe
6308	Invalid axis in ragid tapping or drill cycle command	An invalid axis is specified in G83~G89 command	Refer to param 1022 for axis-property setting, or modify the programe.
6309	Tapping aixs or drilling aixs changed in tapping	Command G84/G88 when tapping， or switch G83/G87, G85/G89 command in drilling cycle.For example:Specify G87 when in G83 state, or specify G88 when in G84 state, or specify G89 when in G85 state	Modify the programe.
6310	Plane changes while tapping	The plane shifting commands G17~G19 are specified in rigid tapping mode.	Modify the programe
6311	Tapping distance too short in ragid tapping	In rigid tapping command G84/G88, tapping distance (distance from R plane to bottom of hole) is less than lead	Modify the programe
6312	Cutteng depth less than retraction depth in deep-hole tapping	In peck tapping command(Q is not set to 0),Value of Q is less than retraction depth setting in Param No.5213	Modify the programe of param
6313	Unusable data specified in tapping	Specify other M code or S code between rigid tapping M code block and G84 block	Modify the programe
6314	Illegal M code specified in tagid tapping or drilling cycle	The M code without sharing a same block is specified in the G83~G89 blocks or in its modal. The M code can be shared with a same block: 1. G83/G87 and G85/G89 are the C axis clamping M code (parameter 5110 setting	Modify the programe

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		value). 2. The M codes of G84/G88 can be shared with a same block: Specify the rigid tapping mode M code (Parameter 5210 setting value; the M code is M29 if the parameter is set to 0); C axis clamping M code (Parameter 5110 setting value).	
6315	M code to clamp C axis error in drilling cycle	M code to clamp C axis error,referring to Param No.#5110, and M30 can't be used	Modify the programe
6316	G84/G88 tapping specified in G96 mode	G84/G88 tapping can't be specified in G96 mode	Modify the programe
6317	Illegal address specified in ragid tapping or drilling cycle	In G83~G89 mode, G7.1/G107, G12.1/G112, G13.1/G113 is specified, alternatively, the polygon machines the G51.2 command	Modify the programe
6318	Illegal T code specified in ragid tapping or drilling cycle	Illegal T code is specified in G83~G89	Modify the programe
6330	Improper command in custom macro program	A function which can't be used in custom macro program is commanded	Modify the programe
6331	Brackets not match in custom macro program	The '['AND']' does not match in the user macro program	Modify the program
6332	Condition command error in custom macro program	Condition comman doesn't exist in custom macro programe.	Modify the program
6333	Format error in macro program	There is an error in other format than <Formula>	Modify the program
6334	Illegal variable number in macro program	A value not defined as a variable number is designated in the custom macro	Modify the program
6335	Unallowable macro program call	A program in G66 modal specified M98, G65 or G66	Modify the program
6336	The nesting of bracket exceeds	The nesting of bracket exceeds the upper limit(5 quintuple)	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	the upper limit		
6337	Illegal argument	The SQRT argument is negitive, the arguments BCD and BIN are negative or BIN argument value can not be shifted into the correct BCD code.	Modify the program
6338	Divided by zero	Divisor was 0(including tan90°)	Modify the program
6339	Quadruple macro modal call	A total of four macro call and macro modal calls are nested	Modify the program
6340	Macro control command cann't be used in DNC and MDI program	Macro control command was specified in DNC and MDI mode	Modify the program
6341	Missing end statement	DO-END does not correspond to 1：1.\nOr has other illegal cmd exists in END block,incorrect format	Modify the program
6342	Substution statement of custom macro not allowed	User's authority is too low to execute subtution statement of custom macro	Modify the program
6343	Illegal loop number	in DOn, 1≤n≤3 is not established	Modify the program
6344	NC and macro statement in same block	NC and custom macro coexist	Modify the program
6345	Illegal macro sequence number	The sequence number specified in the brach statement was not 1~99999, or, it can't be searched	Modify the program
6346	Illegal argument address	An unallowable argument address was used which is not in <Argument Designation>	Modify the program
6347	Tool radius direction data error	The custom macro data used for tool radius direction input should be in the range of 0~9 after rounded	Modify the program
6348	Illegal argument	The argument is incorrect, or the argument is illegal	Modify the program
6349	Operand of logical operation statement error	Operand of logical operation statement OR, XOR, AND are negative.	Modify the program
6350	G67 custom macro	G67 was commanded while	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	canncel	corresponding G66 command was not found. Please check the program whether G66 should be added, and refering to para 6000#1 G67.	
6351	Macro varables are protected	The macro varable is protected from modification	Modify the program
6352	Macro varables modification is forbidden	The macro value is read-only	Modify the program
6353	Overflow of float data	Float data exceeds the allowed range in macro caculating(±1E47 when ARA.6008#0 is set to 1, or else is ±1E308)	Please modify the programe or PARA. 6008#0
6354	Macro prog. shouldn't be called by M code	Macro prog. shouldn't be called by M code	Modify the program
6355	Null value not allowed for system macro	Null value is not allowed to set to system macro	Modify the program
6356	Not proper T code to call custom macro programe	Not proper T code to call custom macro programe	Modify the programe, or refer to Param No.6008#5, 6060~6069
6357	Illegal G code in same block with G66/G67	G66/G67 doesn't work with G code of other group in same block.	Modify the program
6370	No solution at NRC	A point of intersection can't be determined for tool nose radius compensation.	Modify the program
6371	Not allowed to start & cancel NRC in arc comand	Start or cancel tool nose radius compensation in circular interpolation	Modify the program
6372	Can't change plane in NRC	The offset plane is switched in tool nose radius compensation	Modify the program
6373	Interference in circular block	The arc start point or end point coincides with arc center, or destination point is not on arc.	Modify the program
6374	Interference in G90 or G94 block	Overcut will occur in tool nose radius compensation in canned cycle G90 and G94	Modify the program
6375	Interference in arc concluded from	Overcut is possible to occur in tool nose radius compensation	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	checking		
6376	Inconsistent of direction of tool path in NRC and on drawing	Inconsistent of direction of tool path in NRC and on drawing(if exceeds range between 90 and 270 degree)possibly result in part overcut	Modify the program
6377	G41 or G42 not allowed in MDI mode	G41 or G42 was specified in MDI mode(tool nose radius compensation),refering to para 5008#4 MCR	Modify the program
6378	Inner whole circle cutting overcut	In inner whole circle cutting, overcut possibly occur, refering to para 5008#5 CNF	Modify the program
6379	undercut in machining step being less than tool radius	undercut in machining step being less than tool radius, search 5008#6 CNS	Modify the program
6380	Radius of arc is less than that of tool in inner surface arc cutting	Radius of arc is less than that of tool in inner surface arc cutting, which might cause overcut	Modify the program
6381	Arc cmd exists when cancel temporarily or create NRC	While NRC is canceled temporarily as a result of a non-NRC G code, an arc command was specified	Modify the program
6382	Over tolerance of radius in tool compensation calculation	In the cutter compensation, the radius D-value calculated from arc command exceeds its range.	Modify the programe
6383	NRC detected error	Detect error in tool nose radius compensation. This is due to program or operator	Modify the programe
6384	Tool offset not executed before polar cordinate interplation	The tool offset is not executed of the linear axis in Polar cordinate interplation	Modify the programe
6385	Error found when cancelling cylindrical interplation	Cylindrical interplation can't be cancelled in C tool compensation mode	Modify the programe
6386	Property error for axes of plane of C	Property set error for two axis of plane of C tool compensation	Modify the programe

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	tool compensation		
6400	HF/CNR function or CHF/CNR measurement-progr ame-inputing function disabled	If block contains address',R'or',C',refer param No.8134#2 for setting to 0 2.Please refer to paramter No.3453 and parameter No.8134	Modify the programe
6401	CHF/CNR measurement-progr ame-inputing function Only used in automatic mode.	CHF\CNR measurement-programing function do not work in MDI or DNC mode	Modify the program
6402	Code than G01G02/G03 after CHF/CNR	Improper movement other than G01G02/G03 is specified next to chamfer/corner R block	Modify the program
6403	Code is not G01G02/G03 after CHF/CNR	The block next to the chamfer/corner R is not G01G02/G03	Modify the program
6404	Illegle axis after CHF/CNR	An axis not selected in the plane is specified in the block next to the chamfer/corner R	Modify the program
6405	Progame end block after CHF/CNR\nIn auto-meme	DNC mode,the next block is programe end block after CHF/CNR R specified;Or in MDI mode, the block contaning CHF/CNR R is the last block	Modify the program
6406	CHF/CNR R address specified in the NF block of G71~G76	CHF/CNR R address was specified in the NF block of G71~G76	Modify the program
6407	Plane selection not allowed after CHF/CNR	It's not allowed to specified a command to select axis plane in the block next to the chamfer/corner R	Modify the program
6408	Improper movement after CHF/CNR\nA move distance less than	The movement value along axis in the specified block of chamfering or corner R is smaller than the chamfering value or	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	the value of chamfering , chamfering point is not on the tool track	corner R value, alternatively, the chamfering point calculated does not at the path.	
6409	Data error in CHF/CNR	Invalid data of chamfer/corner R is specified	Please modify the programe
6410	Multiple G04 dwells are specified after the chamfering or corner R	In the block, after specifying the block of the chamfering or corner R, two or more G04 dwell commands are specified.	Please modify the programe
6411	None-motion block after CHF/CNR	Blank block or M/S/T/F block without motion command after CHF/CNR.	Please modify the programe
6412	End position or angel not specified in CHF/CNR measurement-programe-inputing function	For blocks after Address (Aa), coordination or angle value should be specified	Please modify the programe
6413	End position calclates error	In the drawing dimension direct input, the specified angle is less than 1 degree. The E-O-B can not be correctly calculated.	Please modify the programe
6414	Address ',A' specified in the last block in direct drawing dimension programing	In direct drawing dimension programing Address, ',A' specified in the last block, so that the destination of the block can't be auto calculatied	Please modify the programe
6415	Address ',A' specified in NF block of G71~G76 command in direct drawing dimension programing	In direct drawing dimension programing, Address ',A' specified in NF block of G71~G76 command, , so that the destination of the block can't be auto calculatied	Modify the program
6416	Fail to calculate destination point	In direct drawing dimension programing, Fail to calculate destination point	Modify the program
6417	Illegal G code specified in direct drawing dimension programing	Illegal G codes are as follows:\nG code of Group 00(G04 exclueded).G code of Group 01(G00,G01,G32 excluded).\nG code of Group 07(G22/G23	Modify the program

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
		excluded),G code of Group 11(G17~G19 excluded).Group 08(G83~G89 excluded)	
6418	Fail to employ direct drawing dimension programing for G code modal state	G code modal state doesn't not work with direct drawing dimension programing, included G codes are as follows:G7.1 circular interpolation,Group 01(G00,G01,G32 excluded),Group 08 G83~G89	Modify the program
6419	Illegal address in CHF/CNR measurement-programe-inputing function	This may be caused by:More than 2 continuous blocks without motion command specified	Modify the program
6430	Illegal tool group number	Tool life group number is less than 1; alternatively, it exceeds the Max. allowable value set by parameter 6813; when parameter 6813 sets to 0, up to 128 groups can be performed.	Modify the program
6431	Tool group number not found	Tool group number commanded in machining program is not set.	Modify the program or parameter
6432	T code not found	In tool life registeration , a T code was not specified where is should be	Modify the program
6433	Illegal tool life data	The tool life to be set is too excessive or not set. When count with time, refer to Param No.6805#1 for time unit.	Modify the program
6434	Tool life management command not matched	T[][]99 not specified or specified error when using T[][]88\nModify the program	Modify the setting value.
6440	Block sequence NO. not found	In programe restart operation, the sequence NO. is not found	Modify the setting value
6441	Illegal assign of G71~G73 cycle blocks for programe restart operation	In restart operation, the block assigned block is included in NS~NF blocks of G71~G73	Please assign alter start block.
6442	Illegal G code in start block of	Illegal G code is specified in start block of programe restart	Please assign alter start block

Alarm No.	Meaning	Possible Alarm Reason	Troubleshooting
	programe restart operation	operation	
6443	Unallowed restart from threading or ragid tapping block	The restart block contains threading commands(G32, G33, G34), thread cycle(Group A:G92, Group B:G78), thread canned cycle(G76), and raid tapping cycle(G84/G88)	Please assign alter start block
6444	System variables operation found in programe restart search	In programe restart search, operation on system variables is not allowed	Please assign again
6445	,Illegal command specified in MDI mode after programe restart search	Only M, S, T commands will execute in MDI mode after programe restart search	Modify the program
6446	G28/G30 command found in programe restart search	G28/G30 command found in programe restart search, while command the start of programe and has not returned to the ref.point	Modify the program

Appendix 3.2 GS Feed Servo Alarm Remedy

No.	Meaning	Main Reason	Remedy
Err-1	AC current motor speed exceeds the value set by PA23 (refer to the speed upper limit set by parameter PA23)	1. Encoder feedback signal abnormal	Check the motor encoder and its signal connection status.
		2. The specified command exceeds the limit set by PA23.	Check the electronic gear ratio and PA23 setting.
Err-2	Main circuit DC bus voltage excessive	1. Braking resistor is disconnected or damaged.	Check braking resistor and its connection.
		2. Braking resistor is unmatched (resistance value is excessive) Note: Smaller resistance means greater current, which will easily cause damage to the braking pipe of the braking circuit.	A. Change to a new braking resistor whose resistance is matched with the power. B. Reduce the ON/OFF frequency according to actual usage.

No.	Meaning	Main Reason	Remedy
		3. Power supply voltage instable;	Check the power supply.
		4. Internal braking circuit damaged.	Change the drive unit.
Err-3	Main circuit DC bus voltage too low	1. If it occurs when the motor is running, the line of input power is cut off or the connection is improper.	Check the input power line
		2. If it occurs when the motor is running, it means the input power voltage is lower than AC180V.	Check the power voltage
		3. If it occurs when the power is turned ON, it means the braking transistor of drive unit is damaged.	Change the drive unit
Err-4	The value in position difference counter exceeds the setting value (refer to the range set by parameter PA17); (When PA18=0, detects the position difference alarm, when PA18=1, does not detects the position difference alarm)	1. The pulse command frequency is too high or the electronic gear ratio is too large.	Check the command frequency of principal computer; check the electronic gear ratio set by PA12/PA13.
		2. The load inertial is excessive or the drive unit torque is insufficient.	A, Check the setting of motor torque limit. B, Improve the drive unit and motor power. C, Lighten the load.
		3. Motor encoder fault or encoder zeroing error.	A, Check the motor encoder and its connection. B, re-zeroing the encoder.
		4. In position mode, the motor U, V, W phase sequence is wrong.	Correct the connection.
		5. position loop or speed loop gain setting is too small (refer to parameter PA5, PA6, PA9)	Adjust the speed loop or position loop gain.
		6. The valid range of position difference is set too small.	Set the PA17 correctly.
Err-5	Motor overheat alarm; the drive unit	1. No temperature detection device in the motor.	Set PA57=0, shield the motor overheat alarm.

No.	Meaning	Main Reason	Remedy
	detects the overheat alarm signal output by the motor. (when PA57=0, the motor overheat alarm is not detected)	2. The temperature detection device type is different with the one set by parameter PA57.	Set the temperature detection device type correctly by PA57.
		3. Overload leads to severe heat of the motor.	Increase the power of drive unit or reduce the load.
		4. In case of severe load, the start/stop frequency is too high.	Reduce the start/stop frequency, and improve the heat radiation condition.
		5. The temperature detection device in the motor is damaged, or the motor inner fault occurs.	Change the AC servo motor.
		6. If the motor temperature detection signal is normal, the drive unit is faulty.	Change the drive unit.
Err-6	Speed amplifier saturation fault	1. Insufficient motor rigidness due to small torque limitation.	Increase the toque limitation value so as to increase the rigidness.
		2. In speed mode, U, V, W phase sequence is reversed.	Connect the U, V, W correctly.
Err-7	Drive prohibition abnormal	The drive prohibition input end terminals FSTP, RSTP are cut OFF.	A. Check the connection and the 24V power of input point.
Err-9	Motor encoder signal feedback abnormal	1. PA48 parameter setting is erroneous.	According to the matched encoder type, set the PA48 correctly and adjust to the default setting.
		2. The motor encoder signal is poor connectedly or the connection is wrong.	Check the connection and signal line welding status.
		3. Motor encoder signal feedback cable is too long, which reduces the signal voltage.	Shorten the cable length within 30m.
		4. Motor encoder is damaged.	Change the motor or encoder.

No.	Meaning	Main Reason	Remedy
		5. Drive unit fault.	Change the drive unit.
Err-11	Drive unit inner IPM module fault	1. It occurs when the power is ON, and the drive unit is not enabled. It cannot be removed after power-on. A, drive unit fault B, Short circuit occurs when braking resistor terminal is grounding	Remedy for reason A is to change to a new drive unit. Remedy for reason B is to check the correct the braking resistor connection.
		2. It occurs when the power is ON, and the drive unit is not enabled. It is removed after power-on again.	It may be caused by external interference or poor grounding. Check the grounding status and interference source.
		3. It occurs when the power is turned ON, and the drive unit is enabled. It cannot be removed after power-on. A. short circuit occurs among motor power line U, V, W, or between U, V, W and PE. B. Drive unit IPM module is damaged. C. Current sampling circuit of drive unit is cut off.	The remedy for reason A is to change the motor line or the motor. The remedy for reasons B, C is to change the drive unit.
		4. It occurs when the motor is starting or stopping and it can be removed after power-on. A, The default parameter of the motor set by drive unit is wrong. B. The load inertial is too large, the commanded accelerated speed is too large when starting or stopping.	The remedy for reason A is to recover the motor default parameter. The remedy for reason B is to increase the acceleration/deceleration time, lower down the accelerated speed or load inertial.
Err-12	The overload alarms during the operation of the motor	1. Motor overcurrent for a long time	Reduce the loading
		2. Incorrect parameter setting may cause the motor vibration or	Readjust the performance parameter relevant to the

No.	Meaning	Main Reason	Remedy
		abnormal noisy.	motor (Refer to the explanations of the PA15, PA16, PA18 and PA19)
		3. Incorrect PA1 setting may cause the improper motor encoder linear number.	Reset the PA1 based upon the type code of the motor
		4. U, V and W wiring is improper. Its operation with power-on is similar with the Er-27.	AC asynchronism spindle motor can be changed any two phases; permanent-magnet synchromism motor is correctly connected based upon the factory wiring criterion, that is, the brown, red and blue cables will separately connected with the U, V and W.
Err-16	Motor thermal overheat	1. The rated current parameter is set incorrectly.	A. Reduce the load. B. Change to a drive device and motor of higher power.
		2. The motor is running with excess current for a long time.	Set the parameter according to the motor nameplate.
Err-17	Braking time is too long	1. The input power voltage is excessive for a long time.	Apply a power which meets the working needs of servo unit.
		2. The braking resistance is too large. The energy cannot be released during braking, causing the rise of internal DC voltage.	Change a correct brake resistor
Err-18	No braking feedback corresponds to braking enable signal	1. Braking circuit fault;	Change a correct brake resistor
Err-19	The DC bus voltage is excessive, but	1. Braking circuit fault;	Change a correct brake resistor

No.	Meaning	Main Reason	Remedy
	there is no braking.		
Err-20	When the power is ON, EEPROM alarm occurs in the inner driver unit.	1. When the power is ON, the drive unit fails to read the data in EEPROM.	Recover the motor default parameter.
		2. EEPROM chips or circuit board fault;	Change the servo drive unit.
Err-21	Power open-phase alarm	Power open phase occurs.	Check the input power.
Err-22	Encoder zeroing alarm	1. Fail to zero the encoder	Change the encoder and readjust the zero
Err-23	Excessive-big of the current error	Current detection circuit fault Current sensor damaged Controllable power voltage fault	Change the drive unit.
Err-24	The 2nd position input signal by detected the CN3 interface is abnormal	1. Without connecting the 2nd position encoder feedback signal, however, the parameter PA97 sets to 0.	Modify PA97=1
		2. Abnormal spindle encoder feedback signal. (The reason is same as the Er-9 alarm)	Inspect the 2nd position encoder signal wiring, welding and the connector
Err-25	Fail to orient the servo unit	1. No Z pulse signal is detected.	Inspect the feedback input signal connection
		2. The corresponding parameter setting is incorrect or the gain setting is excessive big due to the heavy inertial loading.	Inspect the motor type copd PA1 or relevant gain parameters PA15, PA16, PA18 and PA19.
		3. The spindle encoder is inconsistent with the motor encoder signal A/B phase sequence when the 2nd position inut signal is orientated.	Modify PA101 parameter, the phase sequence changes into same, refer to the PA101 parameter manual
Err-27	Incorrect U, V and W wiring (Synchromism motor is enabled)	The phase-sequence of the servo unit main circuit output U, V and W corresponding to the one of the motor is incorrect.	Freely exchange any of two phases
Err-28	Incorrect parameter of software upgrade	Fail to readjust or memory the parameter after the software is copied or updated.	Recall the default parameter, and the power is turned on again after saving

No.	Meaning	Main Reason	Remedy
			the parameter.
Err-29	Parameter error detected after power-on	Conflict occurs when software upgrading.	Re-write the parameter and turn on the power again.
Err-30	AC input voltage alarms due to excessive high	AC power inputs excessive high voltage, which exceeds the 115% of rated voltage	Adjust the electrified net or increase AC reactor, AC filter, etc. equipments to stable the power
Err-32	Illegal code of encoder signals U,V,W	1. The interface is poorly contacted or the cable is poorly shielded.	Check the encoder interface and shielding line.
		2. Encoder U, V, W signals are damaged.	Chang a new encoder.
		3. Encoder interface circuit fault.	Change to a new drive unit.
Err-33	The main circuit voltage is abnormal at power-on	1. The input power voltage is too low or the fluctuation is too large at the moment of power-on.	Check the input power.
		2. Rectifier is damaged or the soft-start circuit is faulty.	Change the servo unit.
Err-34	Pulse electronic gear ratio	The parameter setting of pulse electronic gear ratio is irrational.	Set the PA29/PA30 correctly.
Err-36	Three-phase main power OFF	1. The three-phase power is OFF.	Check the main power and ensure the three-phase AC220V input.
		2. The power detection circuit is faulty.	Change the drive unit.
Err-37	Alarm occurs when the temperature of radiator is below -20℃.	The environmental temperature is too low.	Improve the environmental temperature.
Err-38	Alarm occurs when the temperature is higher than 75℃.	1. The motor overload running for a long time.	Reduce the load.
		2. The environmental temperature is too high.	Improve the ventilation condition.
		3. The drive unit is damaged.	Change the drive unit.
Err-39	Data read error in sensor mode of absolute encoder	1. PA48 parameter setting error;	Set the value of PA48 according to the matched encoder type of the motor, then, adjust to the default value.

No.	Meaning	Main Reason	Remedy
		2. Encoder feedback signal CN2 is disconnected or poorly connected.	Check the CN2 line connection status.
		3. The absolute encoder is damaged.	Change the motor.
Err-40	Abolute encoder data transmission error	Encoder or encoder cable is being interfered.	Check the servo unit and servo motor grounding
Err-41	Absolute encoder multi-core data error	Absolute encoder multi-core data error	1. Change the encoder 2. Check the grounding
Err-42	EEPROM error read in absolute encoder	1. PA1 parameter setting error.	Set the value of PA1 according to the matched encoder type of the motor, then, adjust to the default value.
		2. When the power is ON, the drive unit reads encoder EEPROM error.	Check the CN2 line connection status.
		3. Motor encoder EEPROM is damaged.	Change the motor.
Err-43	Check error when EEPROM is read	1. PA48 parameter setting error;	Set the value of PA48 according to the matched encoder type of the motor, then, adjust to the default value.
		2. After the drive unit reads the encoder EEPROM, data check error occurs.	Execute the Ab-Set encoder write operation.
Err-44	Encoder single-ring/multi-ring configuration error	PA48 parameter setting error;	Set the value of PA48 according to the matched encoder type of the motor, then, adjust to the default value.
Err-45	Encoder data check error	In sensor mode, data check error occurs when the encoder current position is read.	Check the grounding status.
Err-46	A4Ⅱ encoder overspeed	Motor rotates based upon 6000r/min during the servo unit is	Perform the Ab‐rst, and then turn the power on after

No.	Meaning	Main Reason	Remedy
		power off	cutting off, the alarm is then eliminated.
Err-47	A4Ⅱ encoder single-coil resolution error	Motor rotates more than the speed of 100r/min when the servo unit is power on	Confirm that the adapted motor speed is zero when servo unit is power on.
Err-48	A4Ⅱ encoder single-coil counting error	Encoder is being interfered or damaged	Perform the Ab‐rst, and then turn the power on after cutting off; if it does not eliminate yet, it is necessary to change the encoder.
Err-49	A4Ⅱ encoder internal under-voltage	Battery voltage is excessive low	Change the battery, perform Ab‐rst, and then turn the power on after cutting off. It is note that the system should be established a machine tool coordinate after this alarm is eliminated.
Err-51	Position command frequency is excessive high	Position command frequency is excessive high or electric gear ratio is excessive big.	Reduce the position command frequency or correctly set the electric gear ratio
Err-60	Backup inspection with power-on EEPROM fault alarm	There is no backup parameter; alternatively, the verification of the backup space parameter is incorrect.	Make a backup for parameter again, execute EE‐bA operation
Err-61	The motor relevant parameter during verifying the save and backup areas are abnormal when calling the backup parameters.	When recovering the backup operation EE‐rs, its types are inconsistent, as well the linear number of the motor encoder is different.	Resave the parameter, perform EE‐SEt operation
Err-62	The parameter versions, such as the software, backup and save are inconsistent when the power is turned on.	The software version inspected in the backup area is inconsistent with the current one.	Making a backup for the parameter again, and then perform the EE‐bA operation
Err-63	Synchromism/Asynchromism shifting	The hazard operation is being operated, the control software of	This alarm occurs, it is better to touch the factory

No.	Meaning	Main Reason	Remedy
	alarm	the synchronism and asynchromism motors are shifted.	operators.
Err-101	GSKLink communication mst lost alarm	Poor contact or disconnect of the GSKLink communication cable	Check whether the side of the servo and the CNC side communication cable are connected validly.
Err-102	GSKLink communication broken alarm	Poor contact or disconnect of the GSKLink communication cable	Check whether the side of the servo and the CNC side communication cable are connected validly.
Err-103	GSKLink communication mdt CRC verification error alarm	mdt CRC verification error	The power of the CNC and servo unit will be turned on again, if the fault is still generated, it is better to change the servo unit.
Err-104	GSKLink communication FPGA initialization error alarm	FPGA initialization error	The power of the CNC and servo unit will be turned on again, if the fault is still generated, it is better to change the servo unit.

Appendix