广州数控设备有限公司
GSK CNC EQUIPMENT CO., LTD.

Add: No.7, 1st. Street, Luochong North Road, Luochongwei, Guangzhou, 510165, China
Http://www.gsk.com.cn E-mail:sale1@gsk.com.cn
Tel: 86-20-81796410/81797922 Fax: 86-20-81993683

All specifications and designs are subject to change without notice. Mar. 2010/ Edition 2
Dec. 2010/ Printing 1

HTTP://WWW.GSK.COM.CN E-MAIL:SALE1@GSK.COM.CN

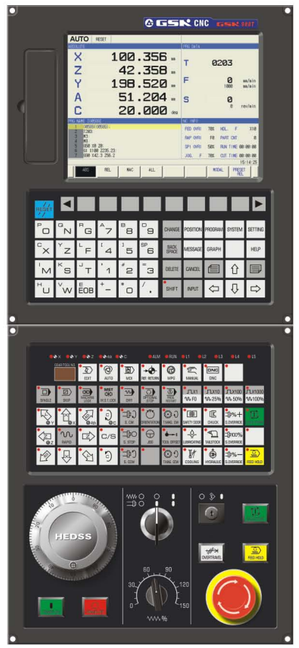
GSK 988T Turning CNC System Volume II Installation and Debugging

GUANGZHOU
CHINA

USER MANUAL

GSK 988T Turning CNC System

(Volume II Installation and Debugging)



广州数控设备有限公司
GSK CNC EQUIPMENT CO., LTD.

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 GSK 988 Turning CNC System made by GSK CNC Equipment Co., Ltd.

This book is User Manual Volume II –“Installation and Debugging”.

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 regarded as unallowable.**

WARNING!

- **Please read this manual and a manual from machine tool builder carefully before installation, programming and operation, and strictly observe the requirements. Otherwise, products and machine may be damaged, workpiece be scrapped or the user be injured.**

CAUTION!

- **Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function configuration and technical performance of a machine tool with this CNC system are determined by machine tool builder's design, so functions and technical indexes are subject to the user manual from machine tool builder.**
- **Though this system adopts standard operation panel, the functions of the keys on the panel are defined by PLC program (ladder diagram). It should be noted that the keys functions described herein are for the standard PLC program (ladder diagram).**
- **For functions and effects of keys on control panel , please refer to the user manual from machine tool builder.**

This manual is subject to change without further 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 information and advice for the users.

User's Responsibility

——Be trained with the safety operation of CNC system and familiar with the safety operation procedures.
——Be responsible for the dangers caused by adding, changing or altering to the original CNC systems and the accessories.
——Be responsible for the failure to observe the provisions for operation, adjustment, maintenance, installation and storage in the manual.

This manual is reserved by end user.

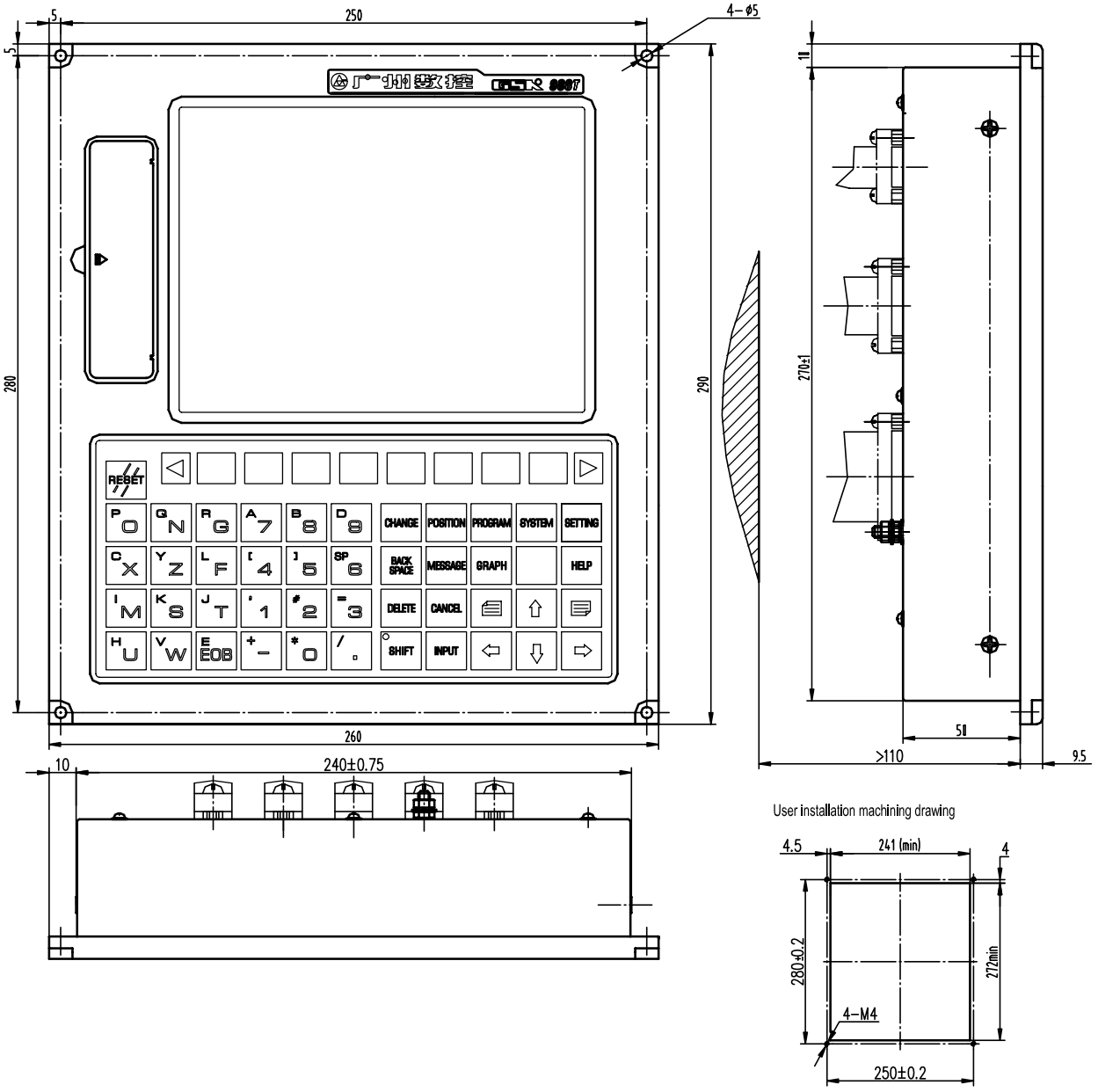
We are full of heartfelt gratitude to you for supporting us in the use of GSK's products.

Contents

CHAPTER I INSTALLATION LAYOUT

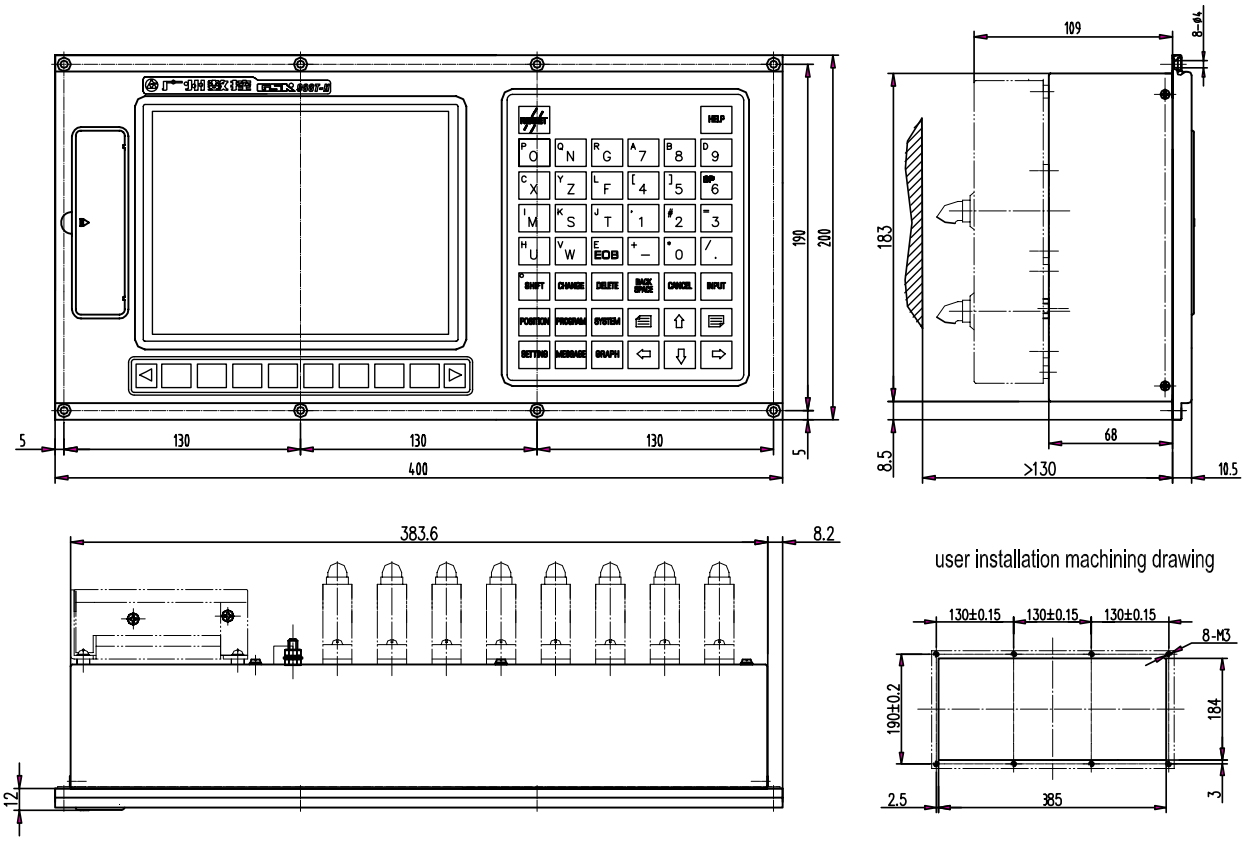
1.1 Overall Dimension of GSK988T and Accessories

1.1.1 Overall Dimension of the GSK988T Mainframe



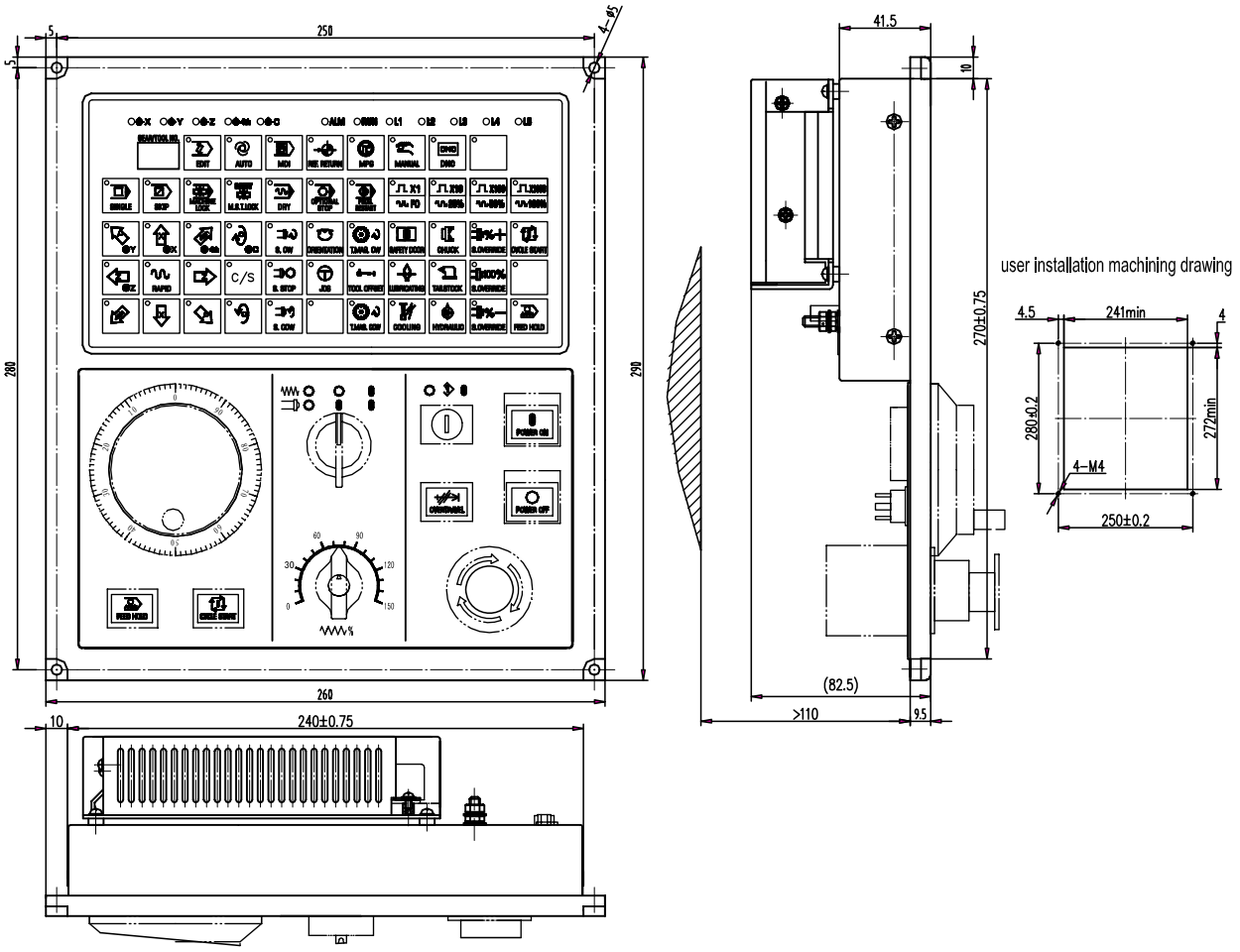
User installation machining drawing

1.1.2 Overall Dimension of GSK988T-H Mainframe

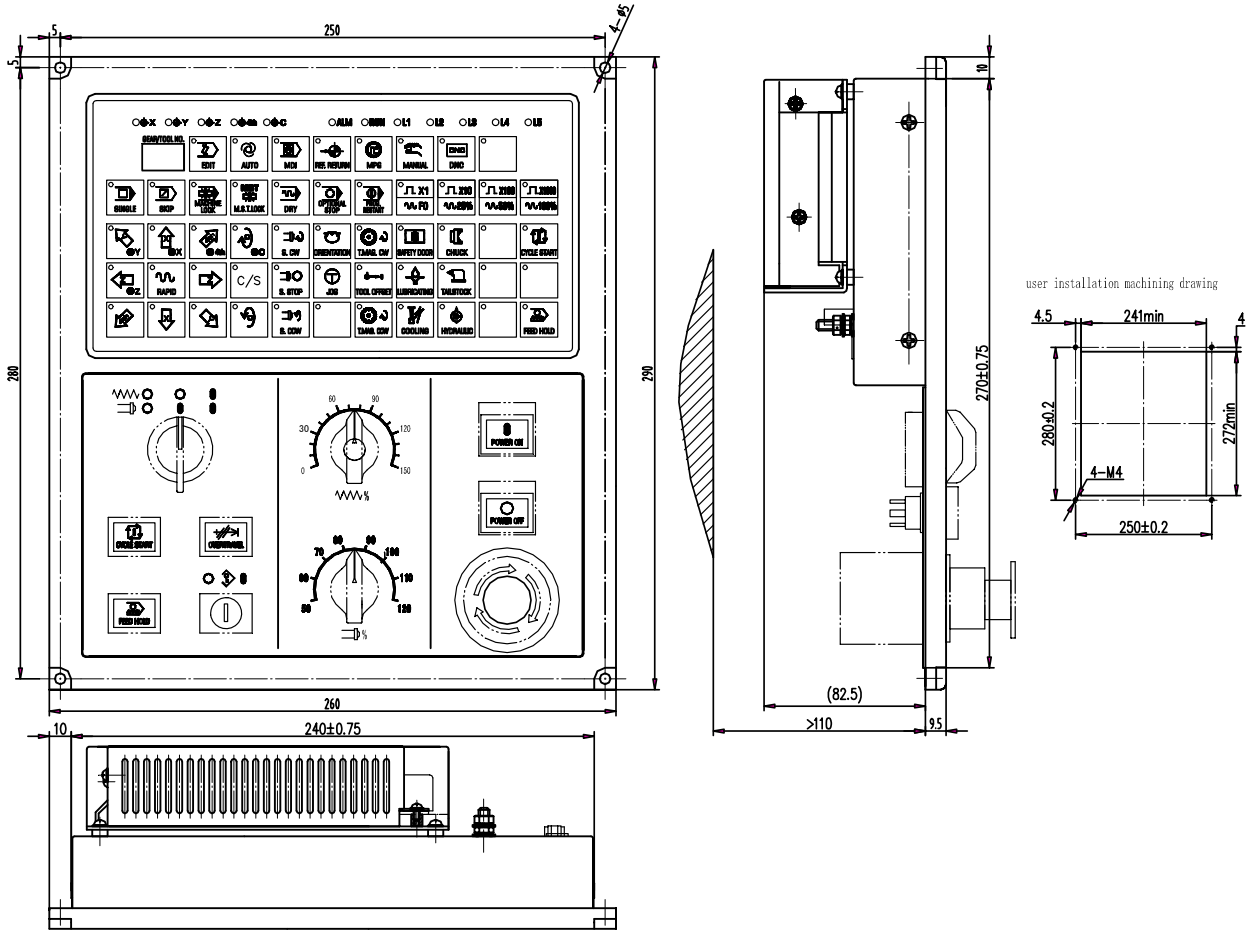


Note: The panel of GSK988T-H is horizontal.

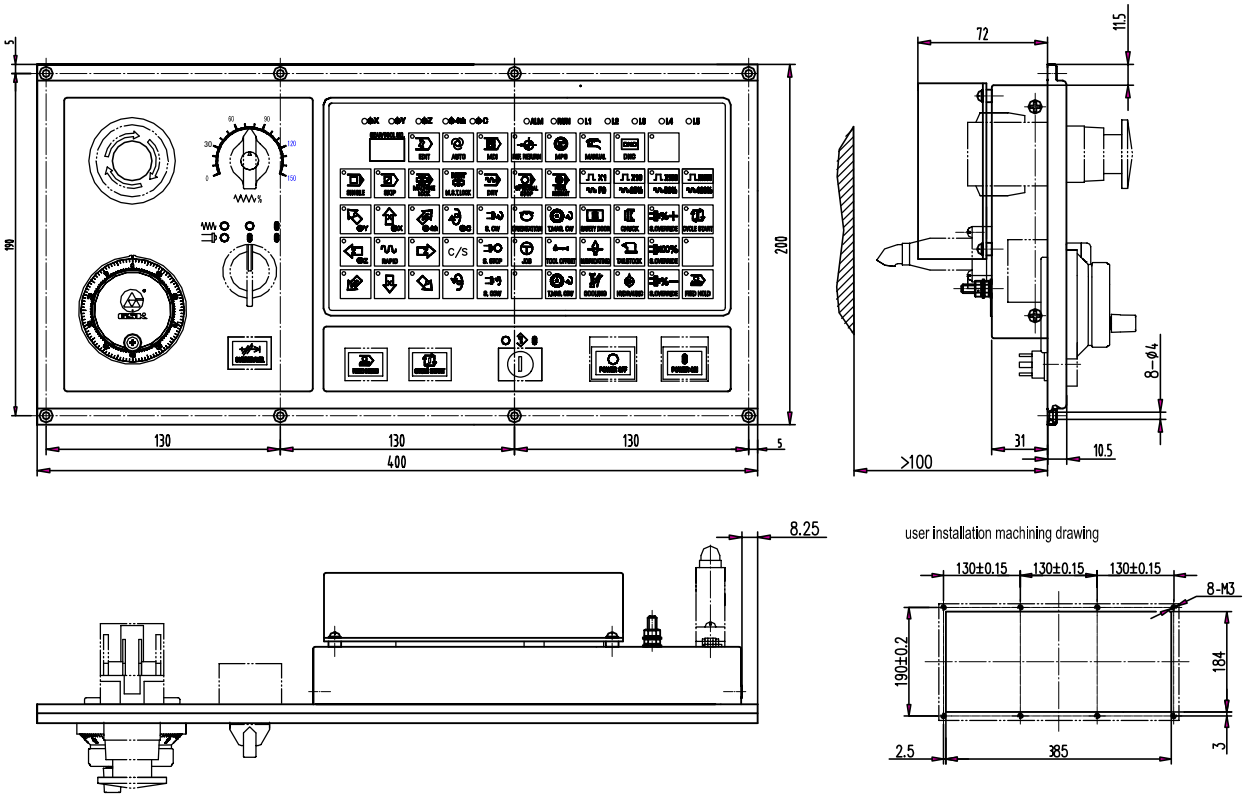
1.1.3 Overall Dimension of GSK988T Operation Panel MPU02A



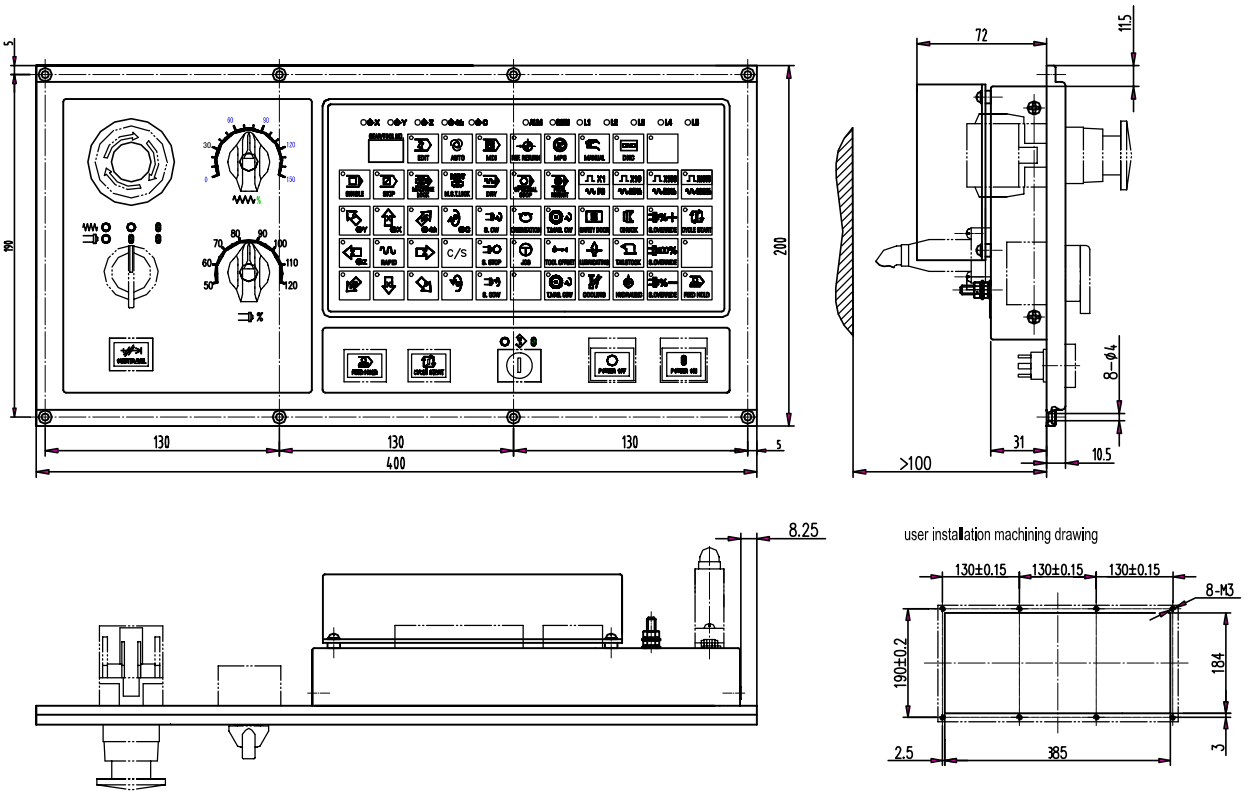
1.1.4 Overall Dimension of GSK988T Operation Panel MPU02B



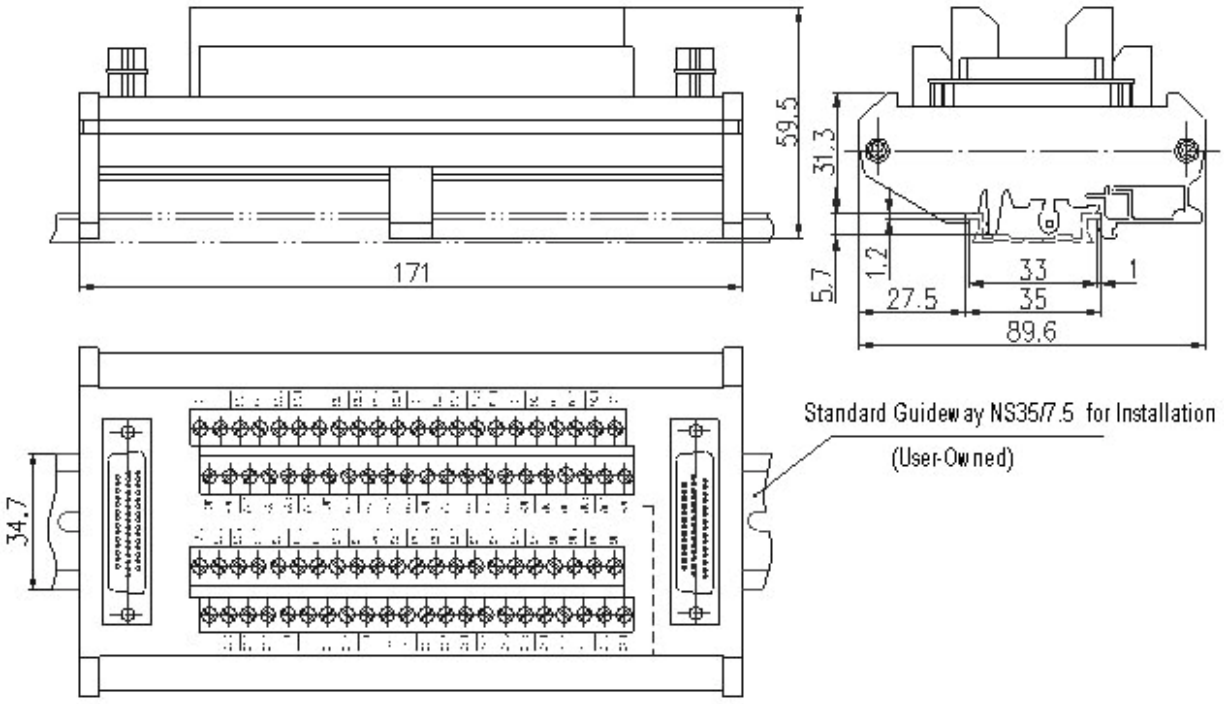
1.1.5 Overall Dimension of GSK988T- H Operation Panel MPU03A



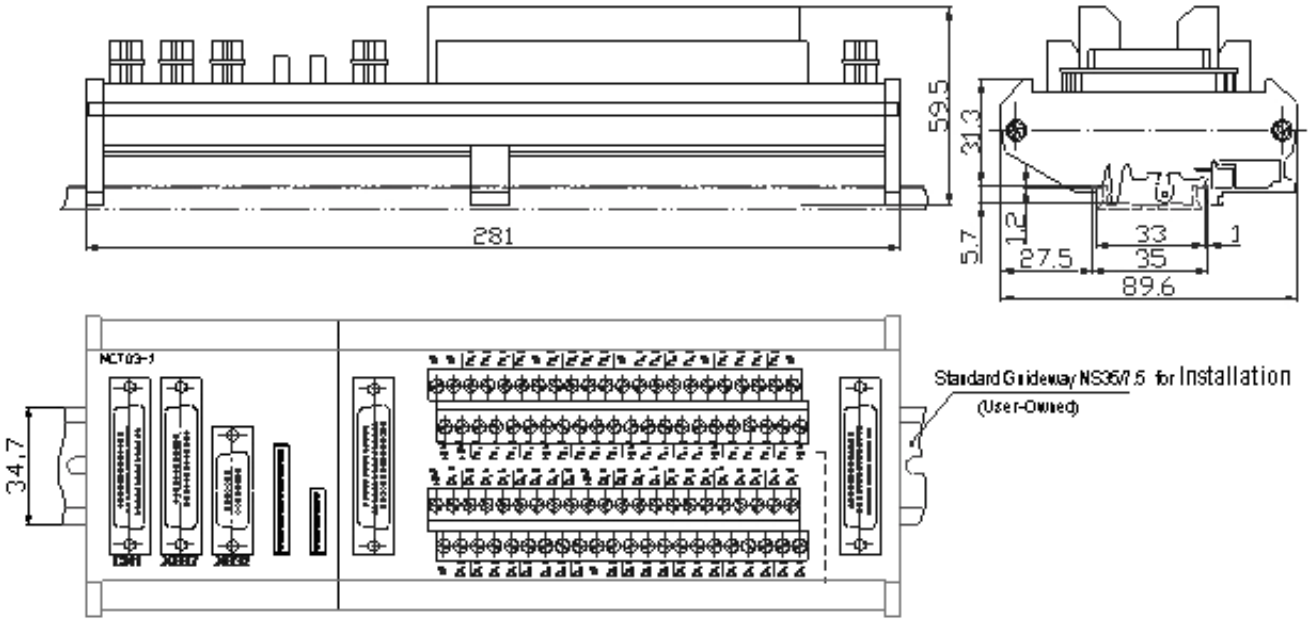
1.1.6 Overall Dimension of GSK988T-H Operation Panel MPU03B



1.1.7 Overall Dimension of I/O Deconcentrator MCT01



1.1.8 Overall Dimension of I/O Deconcentrator MCT02



1.2 Structure of GSK988T Control System

1.2.1 Front /Rear Panel Illustrations

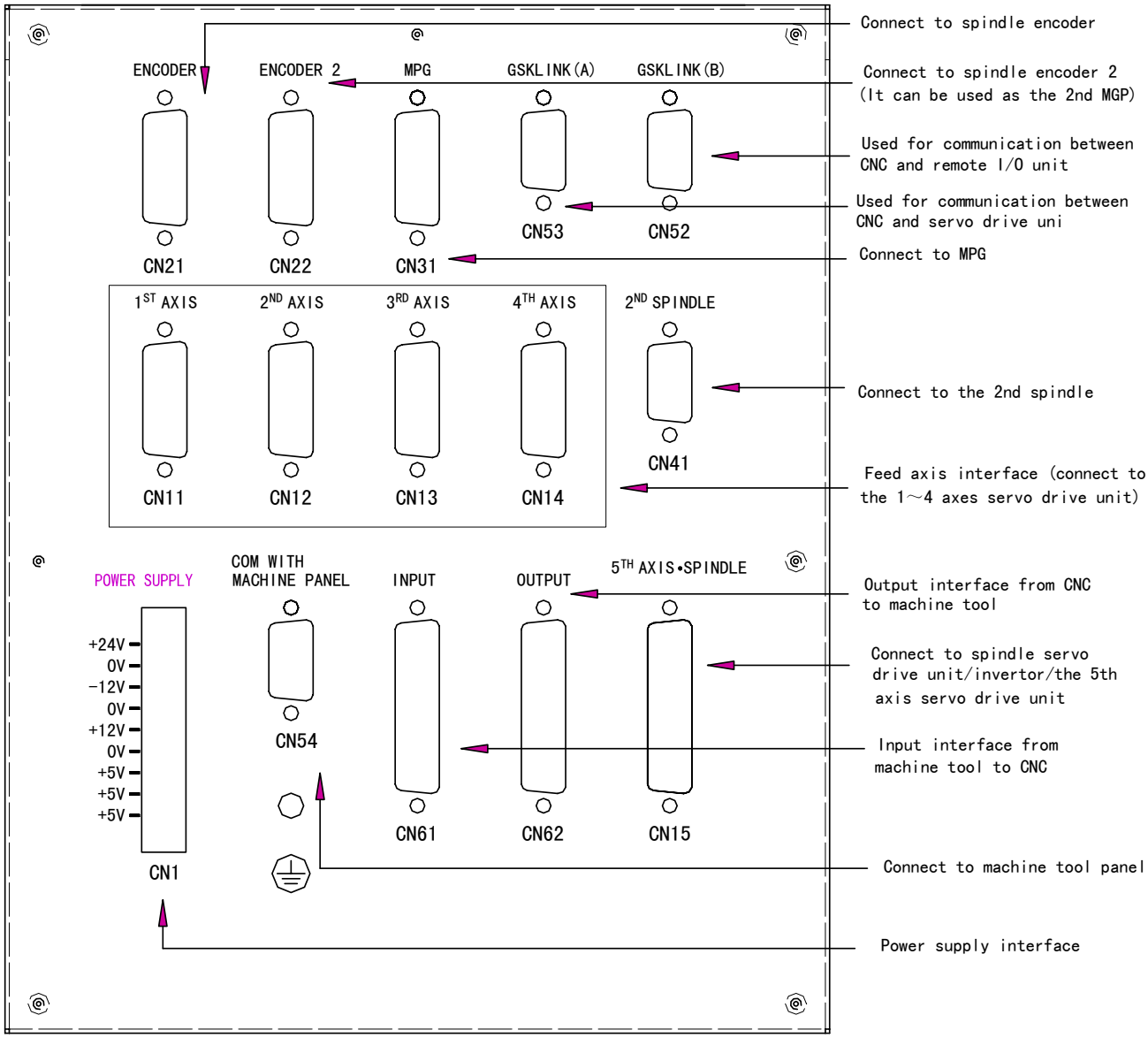


Fig. 1-2-1 The layout of GSK988T mainframe rear cover interfaces

Note: These interfaces are compatible with GSK988T-H system. See Fig.1-2-1

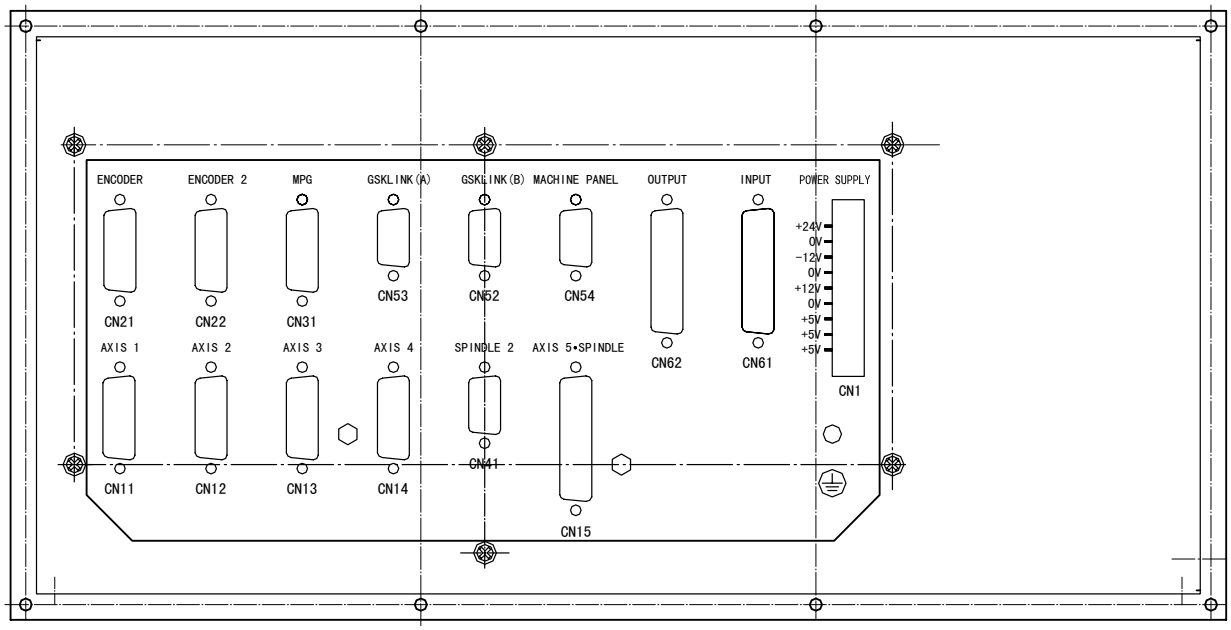


Fig. 1-2-2 The layout of GSK988T-H mainframe rear cover interfaces

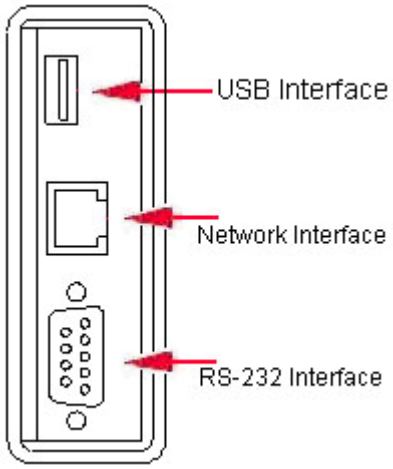


Fig. 1-2-3 The layout of GSK988T front panel interfaces

1.2.2 General Connection Diagram

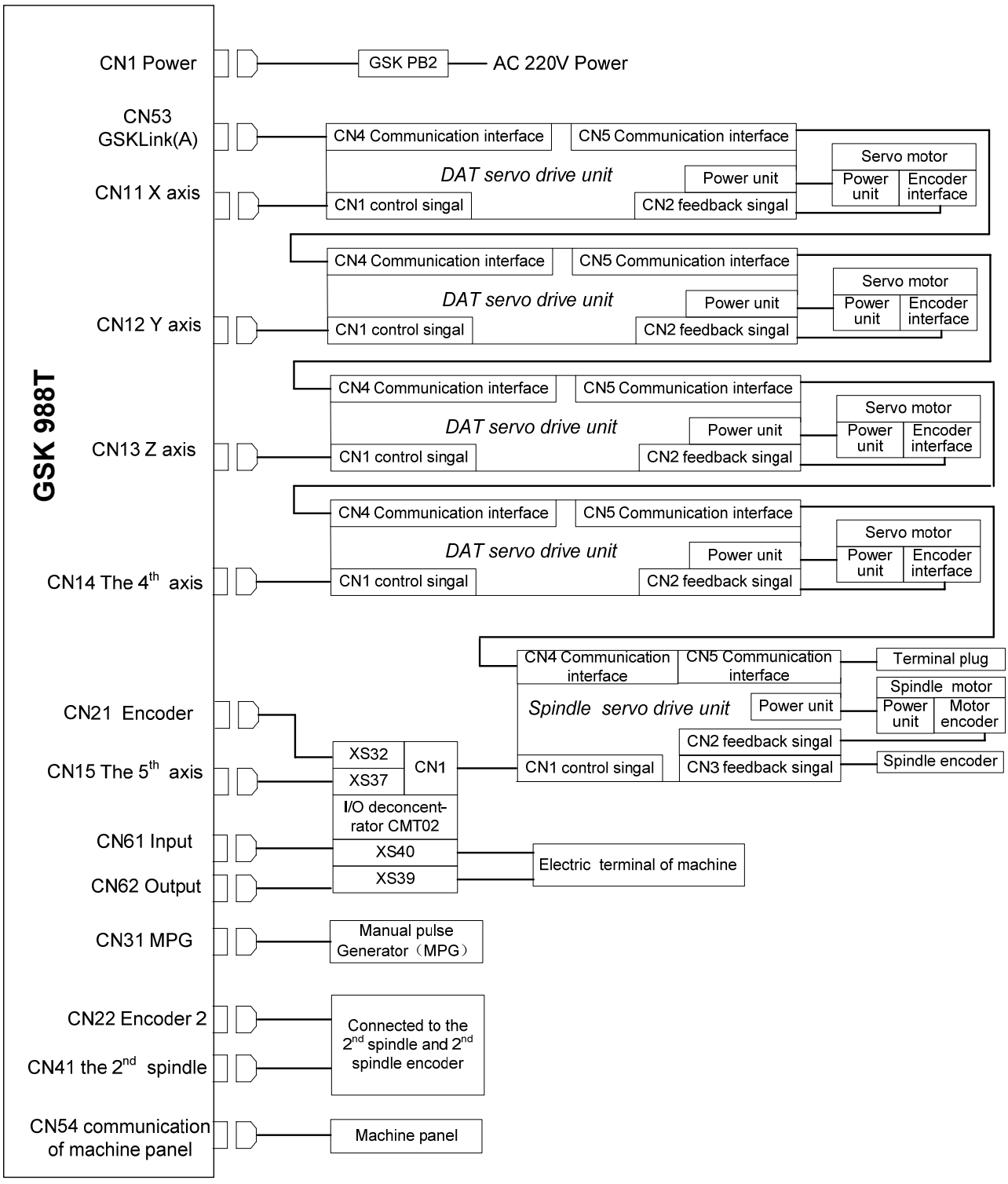


Fig. 1-2-4 GSK988T connection diagram

1.3 GSK988T Installation

1.3.1 Conditions of Electric Cabinet Installation

- Prevent the entry of dust, coolant and organic solution.
- The distance between CNC rear cover and the cabinet should not be less than 20cm. Ensure that the temperature difference (outside and inside the cabinet) will be less than 10℃ in case of temperature rising in the cabinet.
- A radiator fan can be installed inside the cabinet to ensure ventilation.
- The display panel should be installed in proper place to avoid the coolant ejection.
- The interference of external electrical equipments to the CNC should be taken into consideration and be reduced to the greatest extent.

1.3.2 System Grounding Requirements

The following grounding systems are for CNC machine tool:

- Signal ground
  It provides the reference voltage of telecommunication system (0V).
- Frame ground
  It is used for the sake of safety. The shell of frame unit, panel and the interface cables shield should be connected together. It can also suppress the internal and external noise.
- System ground
  It is used to connect the devices and the frame ground with the ground.

Note 1: The connection between signal and frame ground in the CNC control unit is only made at one place.
Note 2: Use the AC power line with grounding wire to ensure grounding during power supply.

1.3.3 Interference Prevention Methods

Measures such as shielding electromagnetic radiation, absorbing impulse current and filtering power noise are taken into CNC design, which, to some extent, protects the CNC to external interference. To ensure a steady working of CNC, it is necessary to take following measures during CNC installation:

① Keep CNC far away from the interference source (such as inverter, AC contactor, static generator, high pressure generator and sectioning for power line, etc.)

② The power to CNC should be supplied via insulation transformer; the machine installed with CNC should be grounding; the CNC and drive unit should be connected with independent grounding wire via grounding point.

③ Interference suppression: connect the RC circuits parallelly at two ends of the AC coil; the RC circuit should be installed to the inductive load as near as possible; fly-wheel diode should be inversely connected in serial at two ends of the DC coil ; surge absorber should be connected in parallel at the winding head of AC motor (see Fig. 1-3-1).

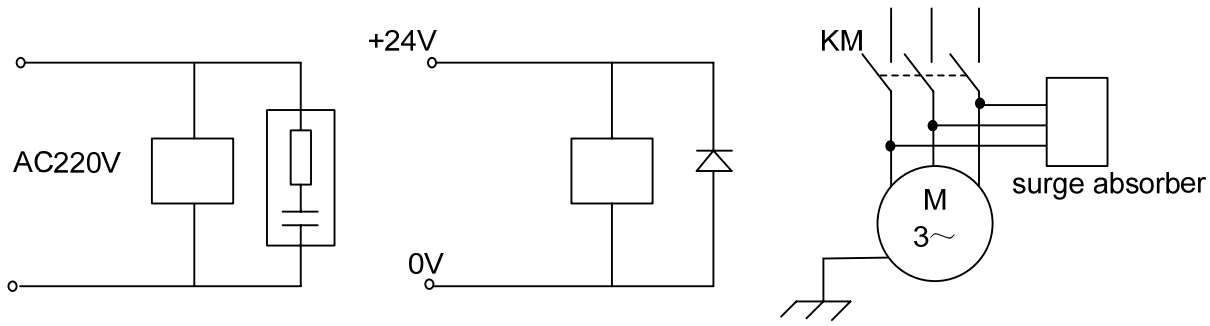


Fig. 1-3-1

④ The outgoing cable of CNC is twisted shielded cable or shielded cable; the shielding layer of the cable is single-end earthed at CNC side; the signal line should be as short as possible.

⑤ To reduce the interference between CNC signal cables and high-voltage cable, the following principles should be followed when wiring:

Group	Cable type	Group	Cable type
A	AC power line	B	DC coil (24VDC)
	AC coil		DC relay (24VDC)
	AC contactor		Cables between CNC and high-voltage electric cabinet
C	Cables between CNC and servo drive unit		Cables between CNC and machine tool

Wiring Requirements:

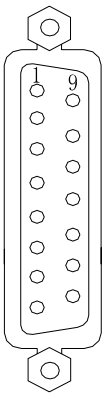
- The cable should be twisted pair.
- Bundle the cables of group A separately from the cables in groups B, C, and the distance should be no less than 10cm; or, make electromagnetic shielding for the cables in group A.
- Bundle the cables of group C separately from the cables in group A, and the distance should be no less than 10cm; or, make electromagnetic shielding for the cables in group C; the distance between group C cables and group B cables should be no less than 10cm.
- Bundle the cables of group B separately from the cables in group A; or, make electromagnetic shielding for the cables in group B; cables in group B should be bundled separately from the group C cables as far as possible.

CHAPTER Ⅱ INTERFACE SIGNAL DEFINITION AND CONNECTION

2.1 Connection with Drive Unit

2.1.1 Definition of the Drive Interface

1：nCP+
2：nDIR+
3：nPC
4：+24V
5：nALM
6：nSET
7：nEN
8：nRDY



9： nCP-
10：nDIR-
11：0V
12：+5V
13：+5V
14：0V
15：0V

Signal	Description
nCP+、nCP-	Command pulse signal
nDIR+、nDIR-	Command direction signal
nPC	Zero point signal
nALM	Drive unit alarm signal
nEN	Axial enable signal
nSET	Pulse inhibition signal
nRDY	Servo ready signal

Fig. 2-1-1 CN11, CN12, CN13 and CN14 interfaces (15 pins, D-type female)

Note: CN1 is the 1st servo axis interface, CN2 is the 2nd one, CN3 the 3rd one, and CN4 the 4th one. Each controlled axis outputs the corresponding servo axis interface, which is set by parameter NO.1023.

2.1.2 Signal Instruction

(1) Command pulse signal and nCP and command direction signal nDIR

nCP+ and nCP- are command pulse signals, nDIR+ and nDIR- are command direction signals, the two groups of signals all are difference (AM26LS31) output, the external is suggested to use AM26LS32 for receiving, refer to the following Fig.2-1-2 about the internal circuit:

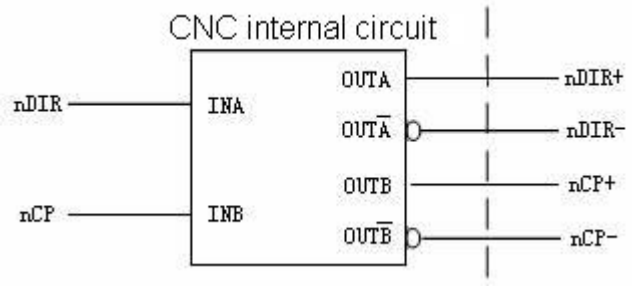


Fig. 2-1-2 Internal circuit of nCP and nDIR

(2) Drive unit alarm signal nALM

The drive alarm level is low or high, which is set by 0 bit of parameter 1816; refer to Fig.2-1-3 for the internal circuit.

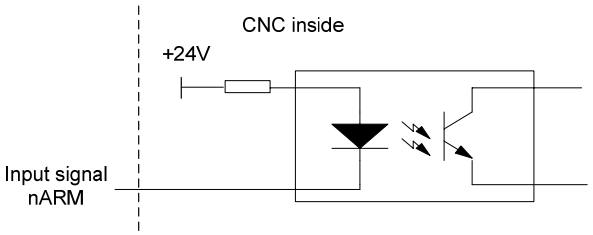


Fig. 2-1-3. Internal circuit of nALM

Input circuit of this type requires that the drive should provide the signal through the methods in the following Fig. 2-1-4:

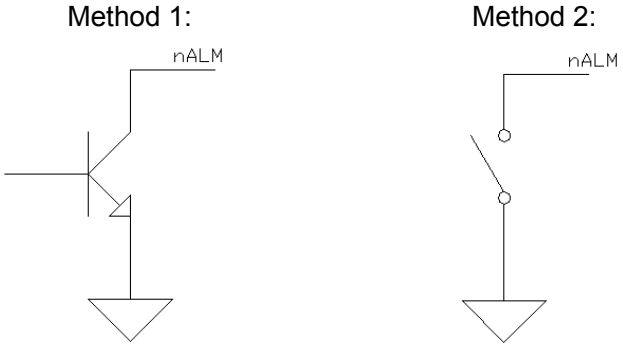


Fig. 2-1-4. Methods of the drive unit providing signals

(3) Servo ready signal nRDY

nRDY signal is connected to the servo drive unit ready signal. See Fig. 2-1-5.

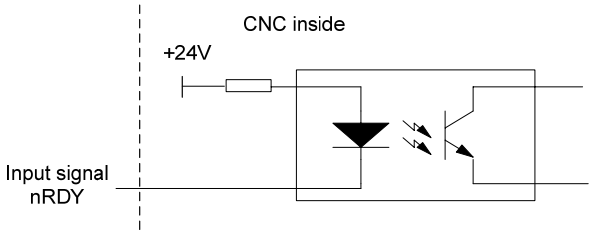


Fig. 2-1-5 Internal circuit of nRDY

(4) Axial enable signal nEN

When CNC is running normally, nEN signal output is valid (nEN signal connects with 0V), and the drive or the emergency stop alarms, CNC switched off, nEN signal outputs (nEn signal cuts off 0V). About the internal interface circuit, refer to the following Fig. 2-1-6:

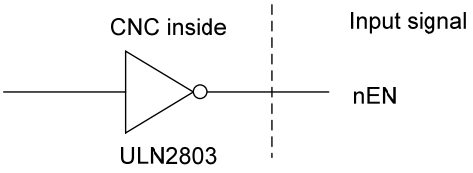


Fig. 2-1-6. Internal circuit of nEN

(5) Pulse inhibition signal nSET

nSET signal indicates the servo input inhibition. To improve the anti-interference ability between CNC and the drive, the signal is low-level when CNC outputs the pulse signal, if there

isn't any pulse signals, it is high level; refer to the following Fig. 2-1-7 about the internal interface circuit:

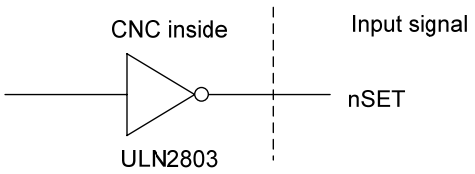


Fig. 2-1-7 Pulse forbidden signal circuit

(6) Zero point signal nPC

Take one-rotation signal of motor encoder or proximity switch signal as zero signals. About the internal connection circuit, refer to Fig. 2-1-8:

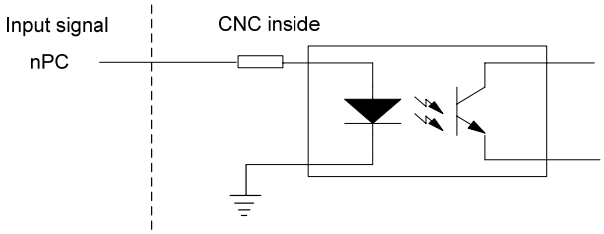


Fig. 2-1-8 Zero point signal circuit

① The illogram of PC signals provided by user is shown in Fig. 2-1-9:

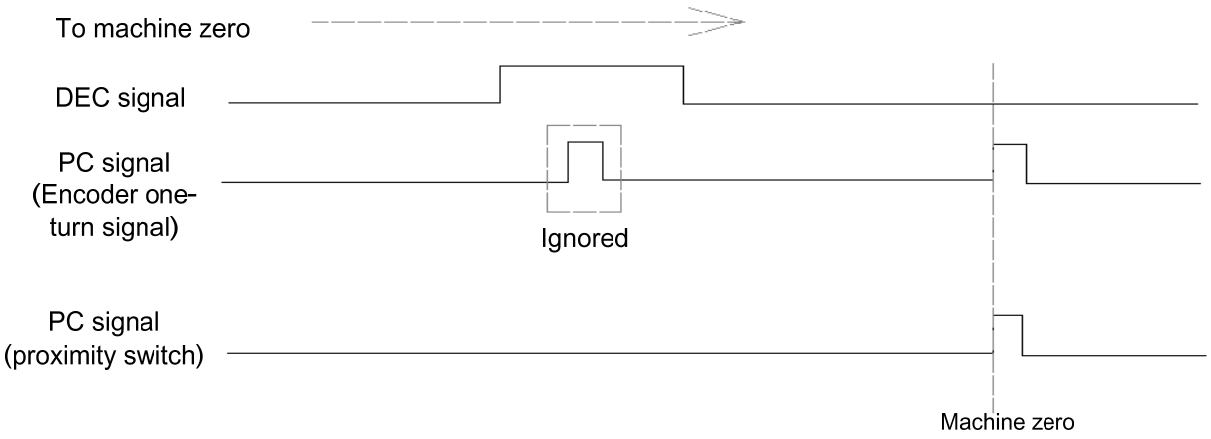


Fig. 2-1-9 Signal illogram

Note: During machine zero return, after releasing the deceleration switch, CNC determines the position of the reference point through detecting PC signal jumping, and the rising edge check and the falling edge check are both valid.

② Refer to Fig. 2-1-10 for the connection method of taking one NPN-type Hall unit as the deceleration signal:

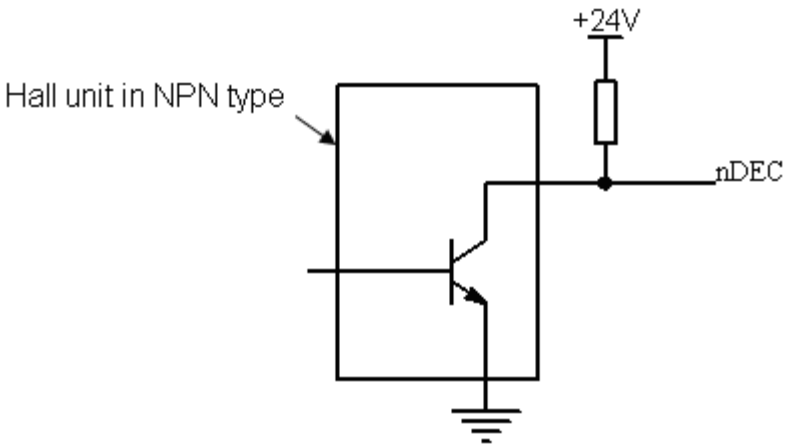


Fig. 2-1-10 Connection with NPN-type Hall unit

③ Refer to the following Fig. 2-1-11 about the connection method of taking one Hall unit in PNP type as one deceleration signal:

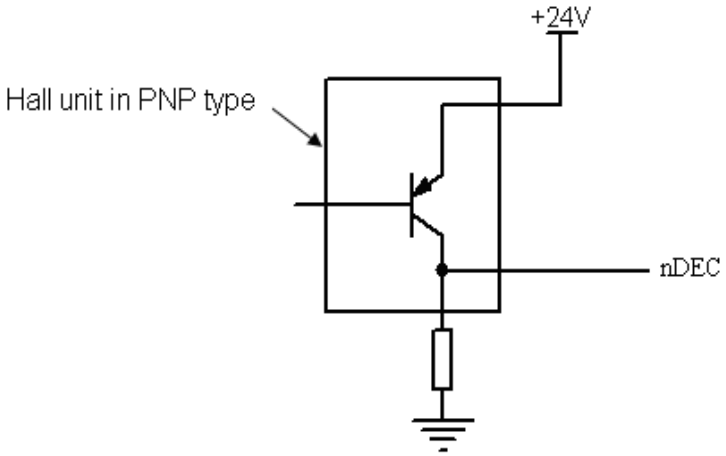


Fig. 2-1-11 Connection with Hall unit in PNP type

2.1.3 Connection with the Drive Unit Interface

The connection between GSK988T system and GSK DA98B drive unit is shown in the following figure.

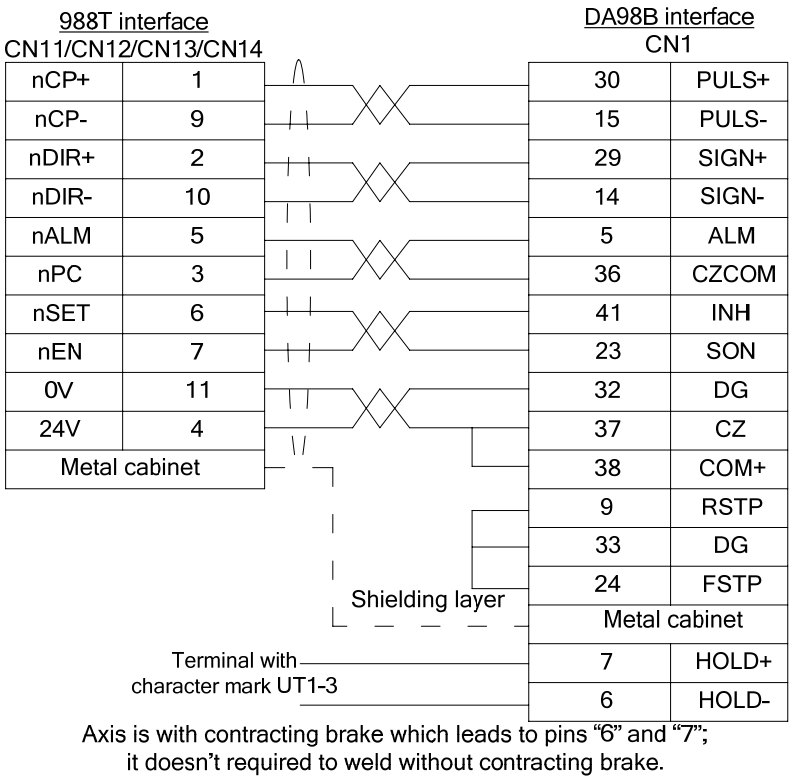


Fig. 2-1-12 Connection between GSK988T and DA98B

The connection between GSK988T and GSK DAT2000C drive unit is shown as follows:

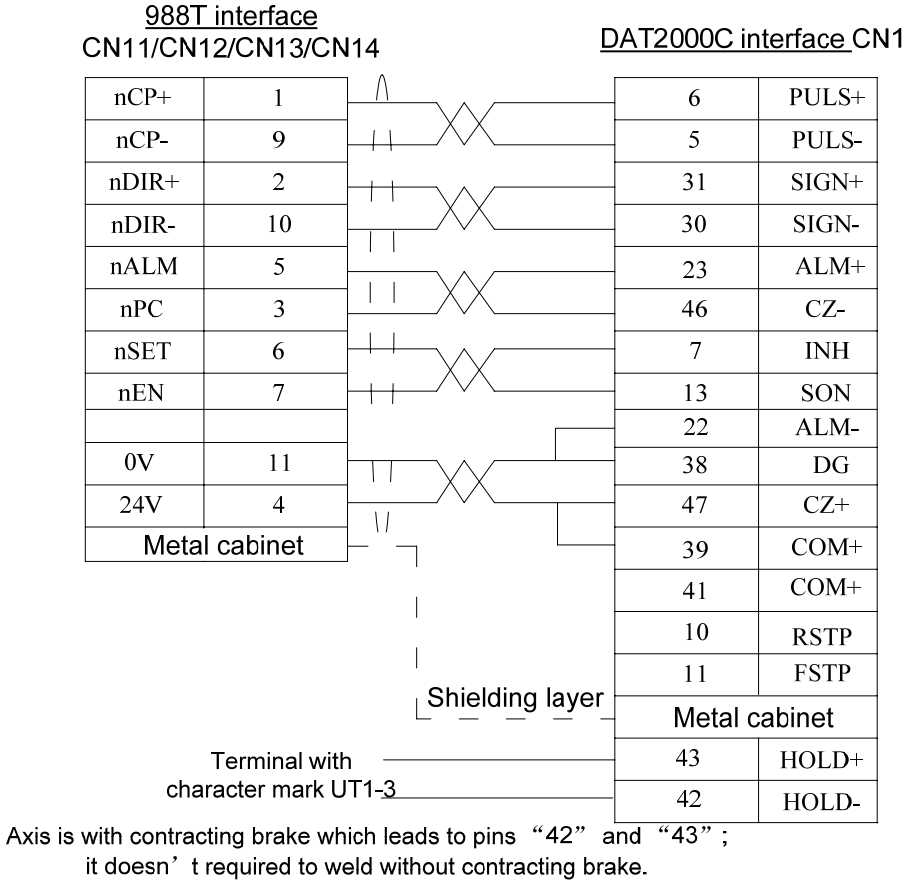


Fig. 2-1-13 Connection between GSK988T and DAT2000C drive unit

2.2 Connection with the Spindle

The spindle interface of GSK988T is CN15 (the fifth axis · spindle interface). It is equipped with the function of pulse output and analog voltage output, and can be adopted with the servo spindle drive unit or the common spindle Inverter, or taken as an independent 5th servo axial interface. Moreover, GSK988T system is also equipped with the 2nd spindle interface CN41 (refer to following chapters for details), and it can output 0~+10V analog voltage for extending the 2nd spindle or the power unit.

2.2.1 The 5th Axis · Spindle Interface Definition

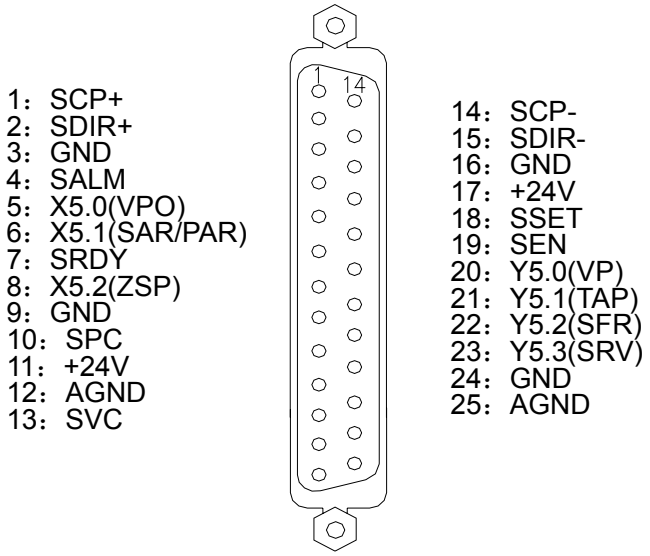


Fig. 2-2-1 CN15 servo spindle interface (25 cords, D type female)

Signal Definition	Explanation	Function Defined by Standard PLC Address
SCP+, SCP-	Command pulse signal	/
SDIR+,SDIR-	Command direction signal	/
SALM	Drive unit alarm signal	/
SRDY	Servo ready signal	/
SSET	Pulse forbidden signal	/
SEN	Axial enable signal	/
SPC	Zero point signal	/
SVC	0~+10V analog voltage output	/
AGND	Analog voltage output ground	/
X5.0 (VPO)	Address of PLC signal, binary input	Spindle speed/position status signal
X5.1 (SAR/PAR)	Address of PLC signal, binary input	Spindle position/speed reaching signal
X5.2 (ZSP)	Address of PLC signal, binary input	Spindle output at zero speed signal
Y5.0 (VP)	Address of PLC signal, binary output	Spindle speed/position switch signal
Y5.1 (TAP)	Address of PLC signal, binary output	Spindle speed loop gain selection signal 2 (used for tapping)
Y5.2 (SFR)	Address of PLC signal, binary output	Spindle CW signal
Y5.3 (SRV)	Address of PLC signal, binary output	Spindle CCW signal
+24V	+24V	/
GND	0V (binary input & output signal ground)	/

2.2.2 Signal Instruction

In the 5th axis • spindle interface, the internal circuits of SCP+, SCP-, SDIR+, SDIR-, SALM, SRDY, SSET, SEN are consistent with that of the similar signal in the drive interfaces CN11, CN12, CN13, CN14. Refer to section 2.1.2.

(1) Zero point signal SPC

SPC signal is valid at low level. It is different with the nPC signal in CN11, CN12, CN13, CN14 interfaces (high-level nPC signal is valid). The internal circuit of SPC is shown in Fig. 2-2-2:

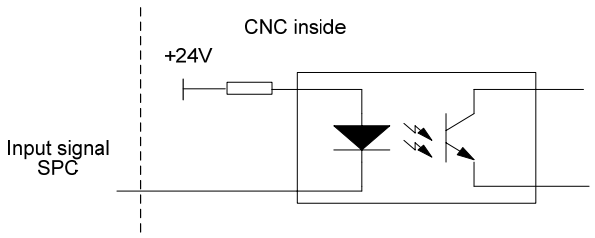


Fig. 2-2-2 Internal circuit of SPC

(2) Signals X5.0, X 5.1, X 5.2, X 5.3, X5.4

Signals X5.0, X 5.1, X 5.2, X 5.3, X 5.4 are the PLC signal addresses; binary input; the internal circuit is shown in Fig.2-2-3.

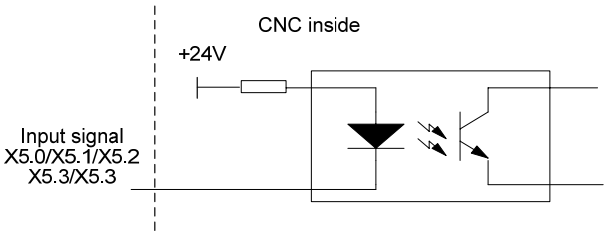


Fig.2-2-3

Note: In the 5th axis • spindle interface, low-level signals X5.0, X5.1, X5.2, X5.3, X5.4 are valid. The X address in general input CN61 (X0.0~X0.7, X1.0~X1.7, X2.0~X2.7, X3.0~X3.7) are valid during high-level input.

(3) Signals Y5.0, Y 5.1, Y 5.2, Y 5.3, Y 5.4

Signals Y5.0, Y 5.1, Y 5.2, Y 5.3, Y 5.4 are the PLC signal addresses; binary output. The internal circuit is shown in Fig. 2-2-4:

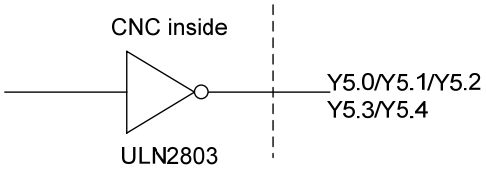


Fig. 2-2-4

2.2.3 Connection with the Servo Spindle Drive Unit

Connection between GSK988T and GSKDAP03C servo spindle drive unit is shown in the following figure. This connection can also be applied in spindle servo drive unit such as GSK DAP03/DAY3025C/DAY3025/DAY3100.

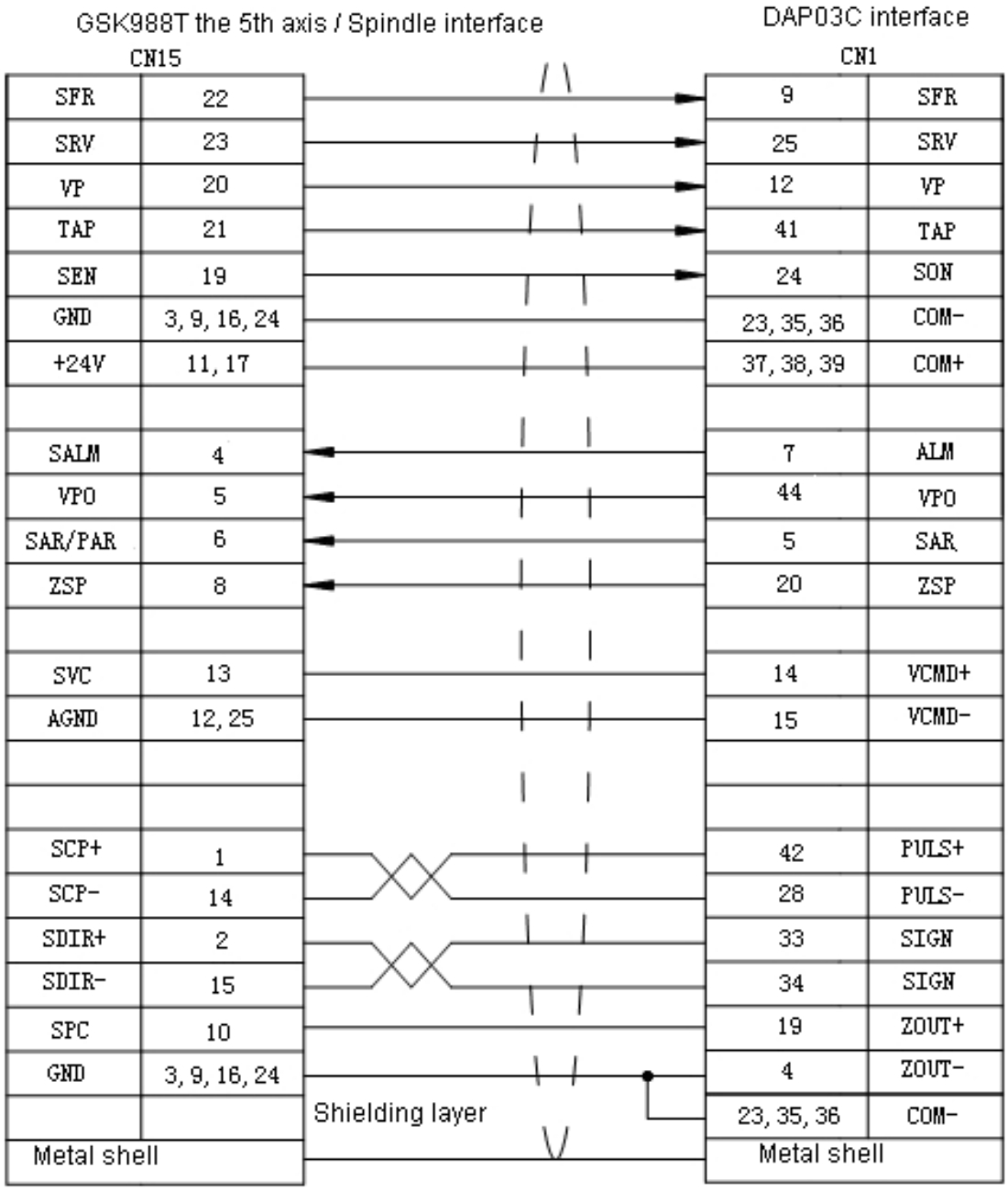


Fig. 2-2-5 Connection between GSK988T and DAP03C drive unit

2.2.4 Connection with the Spindle Inverter Interface

The 5th axis ·spindle interface (CN15) SVC port outputs 0~+10V voltage, the connection between GSK988T and the spindle inverter is shown in the following figure:

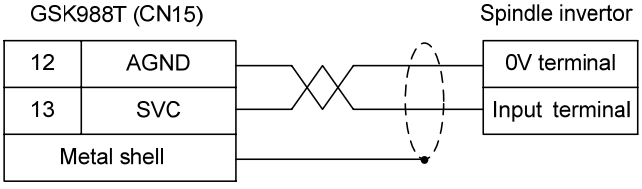


Fig. 2-2-6 Connection between GSK988T and inverter

2.3 Connection with the Spindle Encoder

GSK988T is equipped with two-channel encoder input interfaces (CN21 and CN22), CN21 interface is used as feedback input of spindle speed by default. When multi-spindle control function is started, select the encoder interface which receives the feedback pulse for the system control, through the selection signal PC2SLC (G28.7) of spindle encoder in PLC. When the interface (CN22) of encoder 2 does not connect to the encoder and the selection signal PC2SLC of the position encoder is not set to 1, CN21 interface is taken as the feedback input of the spindle speed.

2.3.1 Interface Definition of the Spindle Encoder

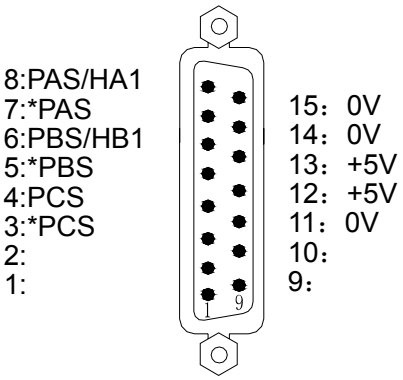


Fig. 2-3-1 Encoder interface of CN21 and CN22 (15 pins, D-type, male)

Signal	Description
*PAS/PAS	Encoder phase A pulse
*PBS/PBS	Encoder phase B pulse
*PCS/PCS	Encoder phase C pulse
HA1 (Only CN22 is with the signal)	The 2nd MPG phase A signal (When it's not used in the 2nd spindle encoder, it can be used to extend the 2nd MPG)
HB1 (Only CN22 is with the signal)	The 2nd MPG phase B signal (When it's not used in the 2nd spindle encoder, it can be used to extend the 2nd MPG.)

2.3.2 Signal Instruction

*PCS/PCS, *PBS/PBS and *PAS/PAS are difference input signals of phase C, B and A respectively; *PAS/PAS and *PBS/PBS is the orthogonal square wave with difference of 90°, the maximum signal frequency <1MHz: The quantity of GSK988T encoder pulses is set by parameter No.3773 (the quantity of the spindle encoder pulses) and No.3803 (the quantity of the 2nd spindle encoder).

2.3.3 Connection with the Spindle Encoder Interface

The connection between GSK988T and the spindle encoder with the twisted pair line is shown in Fig. 2-3-2, and Changchun Yiguang ZLF-12-102.4BM-C05D encoder is taken as one example:

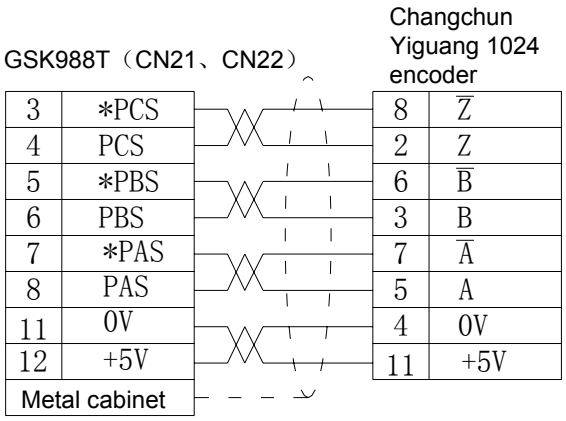


Fig. 2-3-2 Connection between GSK988T and the encoder

2.4 Connection with the 2nd Spindle

988T supports multi-spindle function. Two spindle analog voltage output interfaces include 5th axis ·spindle (CN15) interface and the 2nd spindle (CN41) interface. They are controlled by PLC signals. The 2nd spindle interface can be used to the 2nd inverter spindle or the unit head.

2.4.1 Definition of the 2nd Spindle (Analog Spindle) Interface

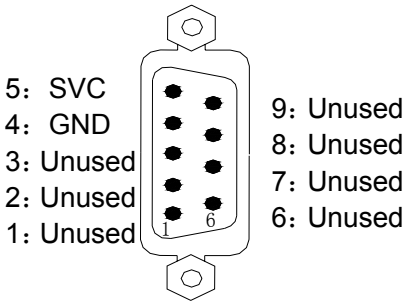


Fig.2-4-1
CN41 analog spindle interface
(9pins, D-type, male)

Signal	Description
SVC	0~+10V analog voltage output
GND	analog voltage output ground

2.4.2 Connection with the 2nd Spindle Inverter Interface

The 2nd spindle interface SVC port outputs 0~10V voltage. The connection is shown in Fig. 2-4-2:

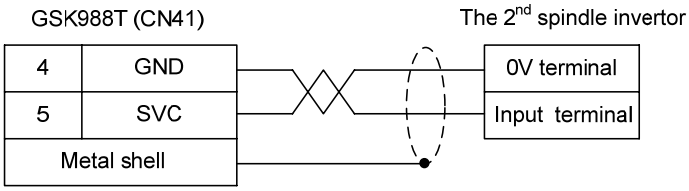
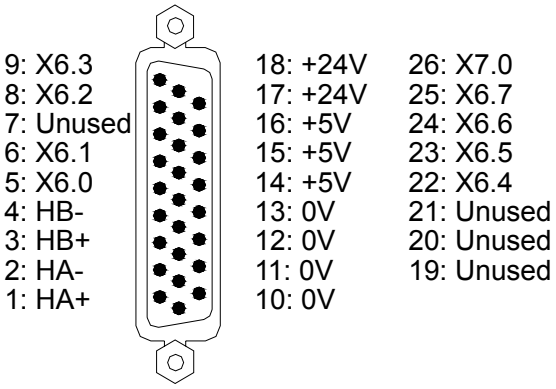


Fig. 2-4-2 Connection between GSK988T and the 2nd spindle inverter

2.5 Connection with MPG

2.5.1 Definition of MPG Interface



Signal	Description
HA+, HA-	MPG phase A signal input
HB+, HB-	MPG phase B signal input
X6.0~X7.0	PLC signal address; binary input

Fig. 2-5-1 CN31 MPG interface
(26 pins, D-type male)

2.5.2 Signal Instruction

HA+, HA- and HB+, HB- are difference input signals of MPG phase A and B respectively. X6.0~X7.0 interfaces are input addresses defined by PLC interface, and it can also be used for axial selection of external MPG box and gear signal input.

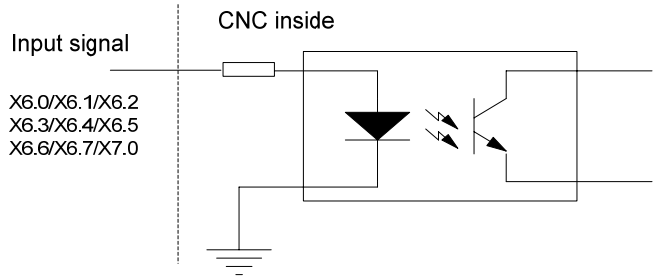


Fig. 2-5-2 Inside circul of X6.0~X7.0 signal

2.5.3 Connection with MPG Interface

The typical connection between GSK988T and MPG is shown as the following figure:

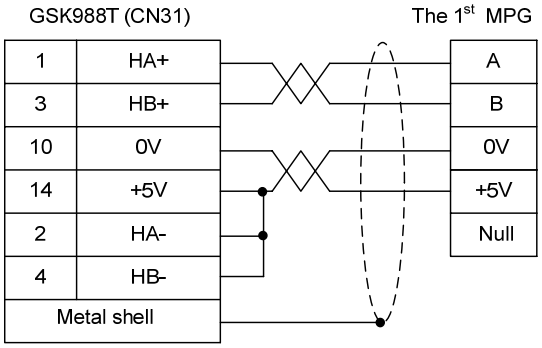


Fig. 2-5-3 Connection between GSK988T and the 1st MPG

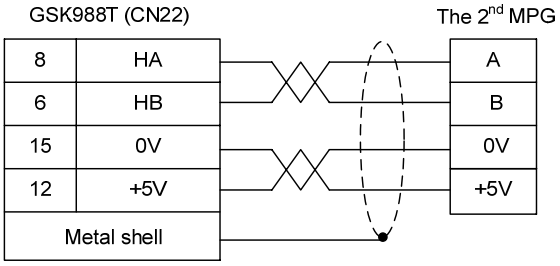
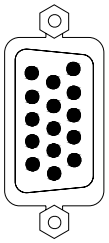


Fig. 2-5-4 Connection between GSK988T and 2nd MPG

2.6 Connection with the Machine Panel

Connect between GSK988T system and the machine panel through communication.

2.6.1 Communication Interface Definition



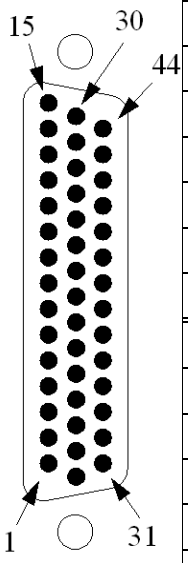
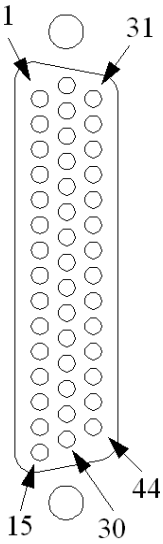
Pin No.	Signal	IN/OUT	Description
1	RXDA	IN	Receive data difference signal
2	RXDB	IN	Receive data difference signal
4	TXDA	OUT	Send data difference signal
5	TXDB	OUT	Send data difference signal
7	RESET	OUT	Panel resetting signal

Fig. 2-6-1 Standard machine panel interface CN54
(15 pins, D-type male)

2.7 GSK988T General I/O Interface Definition

2.7.1 Definition of Input & Output Addresses

Table 2-11 Definition of input & output addresses

Interface	CN61 Pin No.	PLC Address	Interface	CN62 Pin No.	PLC Address
 CN61 (male) input	1	X0.0	 CN62 (female) output	1	Y0.0
	2	X0.1		2	Y0.1
	3	X0.2		3	Y0.2
	4	X0.3		4	Y0.3
	5	X0.4		5	Y0.4
	6	X0.5		6	Y0.5
	7	X0.6		7	Y0.6
	8	X0.7		8	Y0.7
	9	X1.0		9	Y1.0
	10	X1.1		10	Y1.1
	11	X1.2		11	Y1.2
	12	X1.3		12	Y1.3
	13	X1.4		13	Y1.4
	14	X1.5		14	Y1.5
	15	X1.6		15	Y1.6
	16	X1.7		16	Y1.7
	29	X2.0		29	Y2.0
	30	X2.1		30	Y2.1
	31	X2.2		31	Y2.2
	32	X2.3		32	Y2.3
	33	X2.4		33	Y2.4

	34	X2.5		34	Y2.5
	35	X2.6		35	Y2.6
	36	X2.7		36	Y2.7
	37	X3.0		37	Y3.0
	38	X3.1		38	Y3.1
	39	X3.2		39	Y3.2
	40	X3.3		40	Y3.3
	41	X3.4		41	Y3.4
	42	X3.5		42	Y3.5
	43	X3.6		43	Y3.6
	44	X3.7		44	Y3.7
	17	X4.0		17~19, 26~28	0V
	18	X4.1		20~25	+24V
	19	X4.2			
	20	X4.3			
	25	X4.4			
	26	X4.5			
	27	X4.6			
	28	X4.7			
	21~24	0V			

2.7.2 Input Signal

Input signal is the one which the machine electric wire or the machine panel transmits to CNC, and after connecting the input signal and +24V, the input is valid; if they are cut off, the input is invalid. The input signal of contacts on the machine side should satisfy the following conditions:

Contact capacity: DC30V, 16mA above;

Leakage current between contacts during opening: Below 1mA;

Voltage drop between contacts during closing: Below 2V (Current 8.5mA, including the cable potential drop).

There are two methods of external input for input signals: one is switch input with contacts, the connection is shown in Fig. 2-7-1:

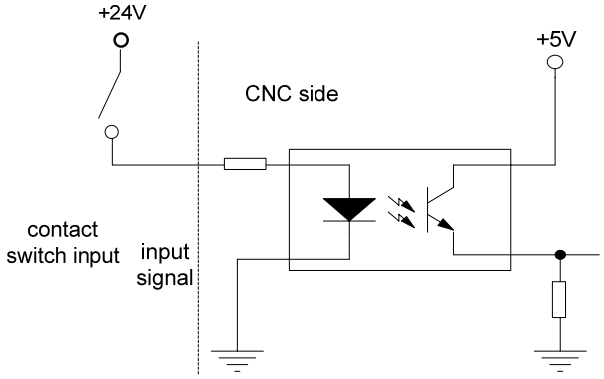


Fig. 2-7-1

The other is switch (transistor) input free of contacts; connection is shown in Fig. 2-7-2 and Fig. 2-7-3.

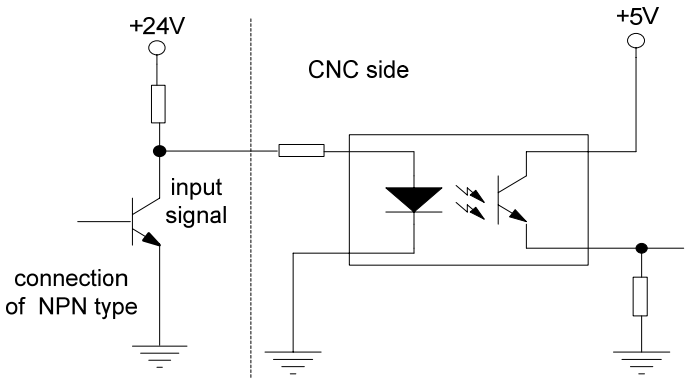


Fig. 2-7-2 NPN type

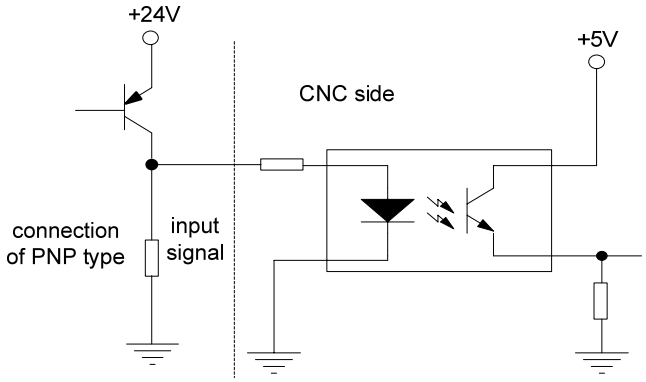


Fig. 2-7-3 PNP type

2.7.3 Output Signal

Output signal is used for the drive machine electrical wire side or the relay and the indicator on the machine panel side. When the output signal connects with 0V, the output function is valid (Y output signal is 1); cut off 0V, the output function is invalid (Y output signal is 0). The circuit is shown in the following Fig. 2-7-4:

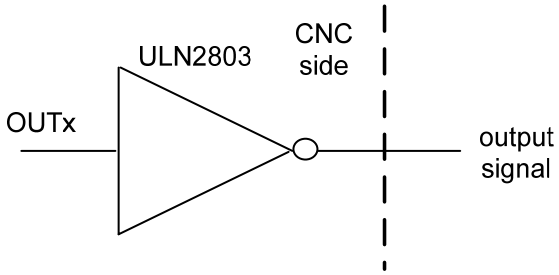


Fig. 2-7-4. Internal circuit of the output signals

Therefore, the signal has two output statuses: OV output or high resistance. The typical application is as below:

- Drive light diode

Use ULN2803 to output drive light diode and need the serial connection with one resistance,

limit the current from light diode (normally 10mA), which is shown in Fig. 2-7-5:

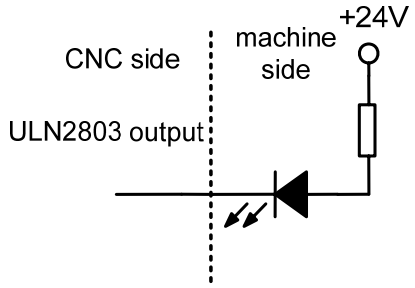


Fig. 2-7-5: Drive light diode

- Indicator in drive filament type

ULN2803 is used to output the indicator in drive filament type, and externally connect with one preheated resistance to reduce the current shock during break-over, and the value of the preheated resistance is based on that the indicator is off, which is shown in Fig. 2-7-6:

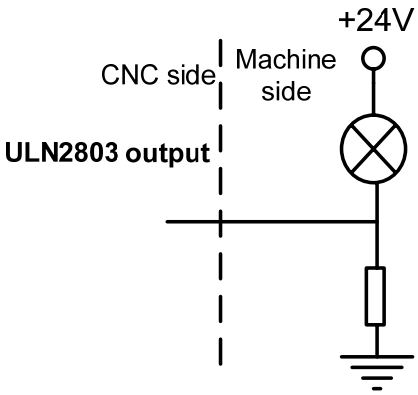


Fig. 2-7-6

- Drive inductive loading (such as the relay)

Output the drive inductive loading in ULN2803 type and it requires connecting the fly-wheel diode close to the circuit, which is to protect the output circuit and reduce the interference, which is shown in Fig. 2-7-7:

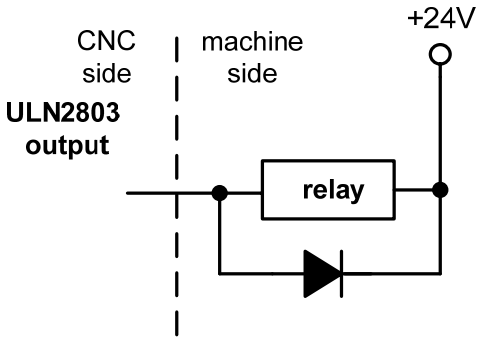


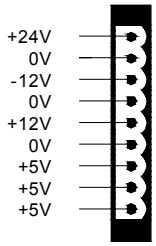
Fig. 2-7-7

2.8 Connection with the Power Supply

GSK988T uses GSK-PB2 power supply box, There are 4 groups of voltage: +5V (3A), +12V (1A), -12V (0.5A) and +24V (0.5A), and common port COM (0V).

2.8.1 Definition of Power Supply Interface

The interfaces of power supply are shown in Fig. 2-8-1 and 2-8-2:



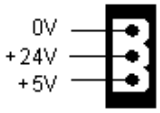


Fig. 2-8-1 power supply interface CN1 Fig. 2-8-2 power supply interface on the panel

2.8.2 Connection between GSK988T and GSK-PB2 Power Supply Box

When GSK988T is dispatched from the factory, GSK-PB2 power supply box and GSK988T power supply interface has been already connected, so the user just need to connect to 220V AC power supply. The connection between GSK-PB2 power supply box and GSK988T power supply interface is shown in Fig. 2-8-3:

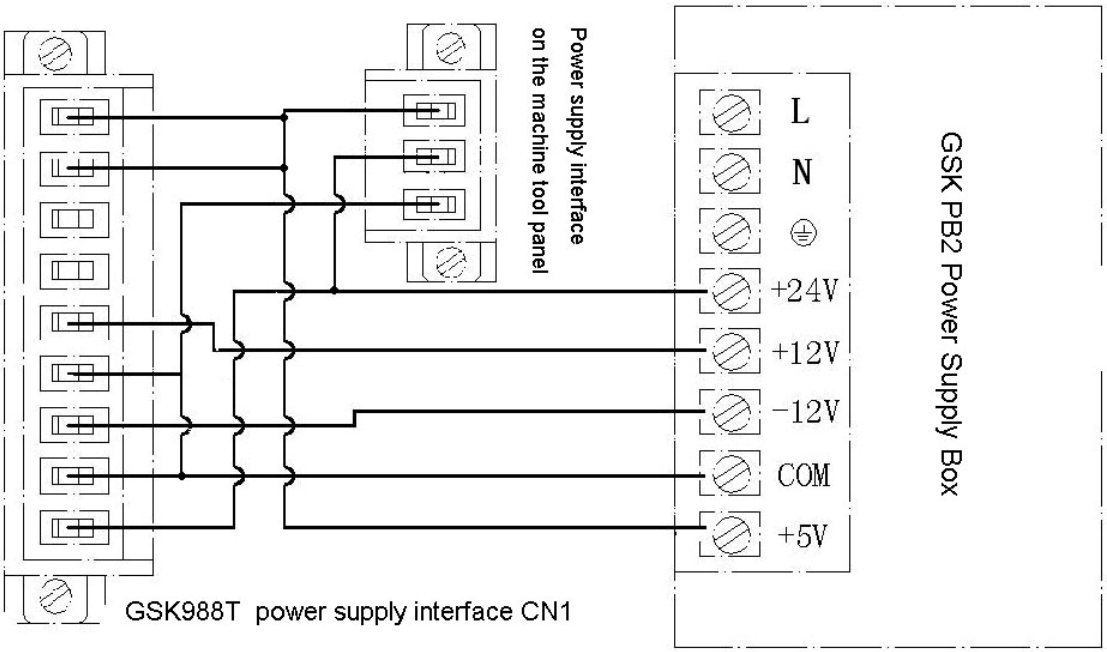


Fig. 2-8-3

2.9 Connection with the External Equipment

There are three interfaces on the left side of GSK988T LCD display screen: USB (flash driver), internet and RS-232 interfaces, which are shown in the following figure. All the three interfaces can be used for processing the file, two-way transmission between the system Para file and PLC file and upgrading the system software. Among them, the internet interface can also be used for remote monitor from PC to 988T system.

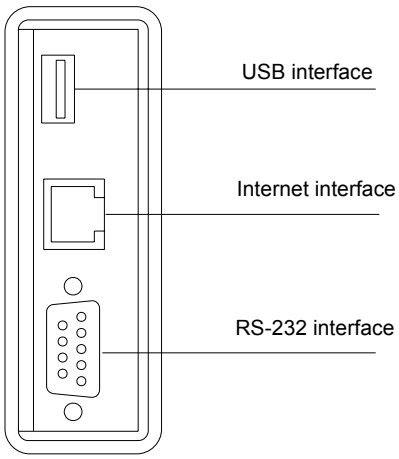
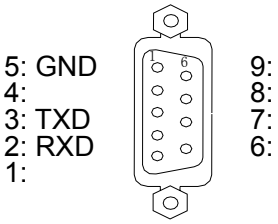


Fig. 2-9-1 GSK988T front panel interface

2.9.1 RS-232 Interface Definition

RS-232 communication interface:



Pin No.	Signal
2	RXD
3	TXD
5	GND

Fig. 2-9-2 RS-232 interface
(9 holes, D type female)

GSK988T executes communication through RS232 with PC (GSKComm-M communication software should be installed). The connection is shown in Fig. 2-9-3:

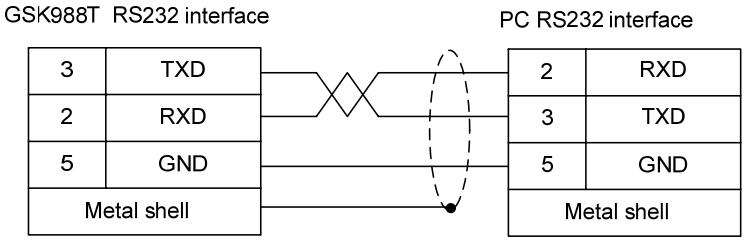


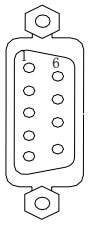
Fig. 2-9-3 Connection between GSK988T and PC

2.9.2 Definition of GSKLINK Bus Interface

GSK988T is with GSKLink interfaces of two routes for connecting with the remote IO units and the servo drive unit with GSKLink communication function. Among them, CN53 (GSKLINK serial bus A) is for communication between CNC and the servo drive unit to realize real-time monitor of servo parameter configuration and servo unit; CN52 (GSKLINK serial bus B) is for

communication between CNC and remote IO unit.

1：GND
2：CANn_L
3：
4：GND
5：



6：
7：CANn_H
8：
9：

Fig 2-9-4 GSKLink bus interface CN53 and CN52 (9 holes, D type female)

Signal	Description
CANn_L	Low level of data difference signal
CANn_H	High level of data difference signal
GND	Signal ground

2.9.2 Network Interface Definition

Network interface (standard):

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	Chassis ground

2.9.3 USB Interface Definition

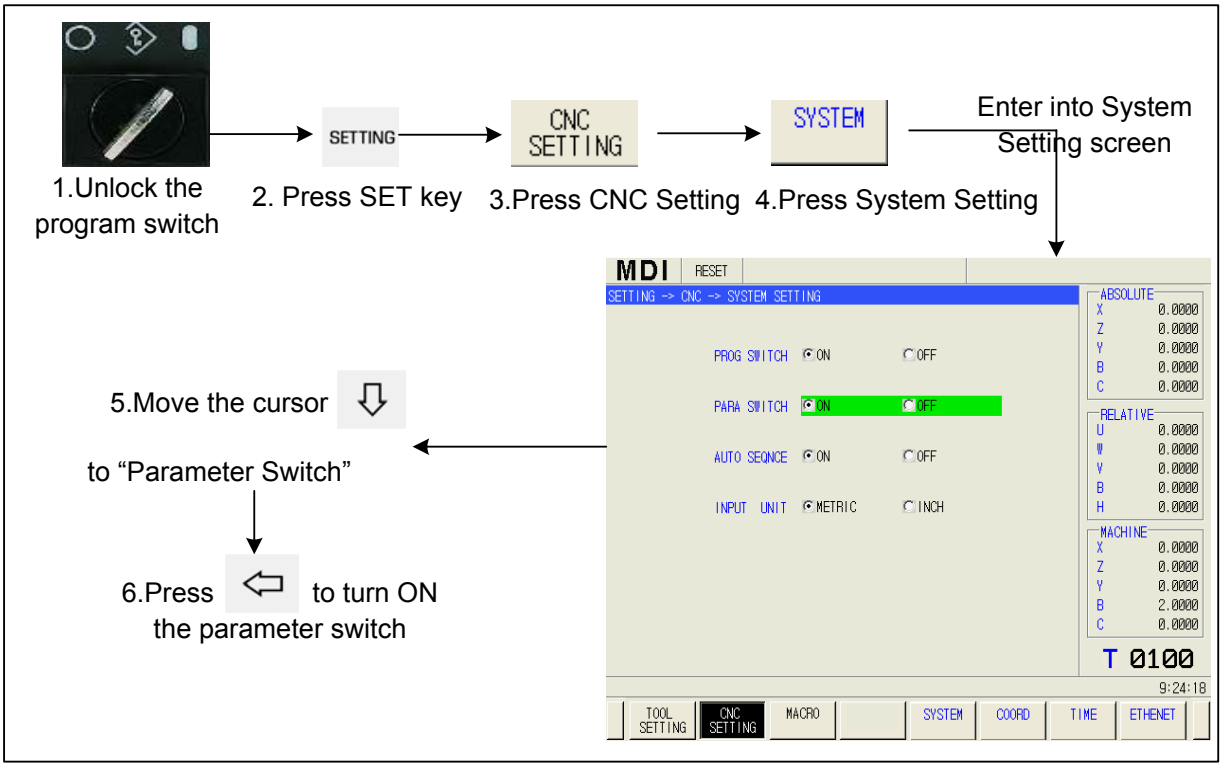
Main USB interface (standard):

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

CHAPTER III MACHINE TOOL DEBUGGING-OPERATION

3.1 Parameter Setting

The modification, backup and recovery of GSK988T system parameters and servo parameters can only be done under such conditions: higher than 3rd management level; parameter switch is ON and the system is in MDI mode. The operation of turning ON the parameter switch is shown as follows:



Note 1: After parameters are modified, the modification is valid to some parameters immediately; some will be valid only after power on again. For details, please refer to chapter 5 Parameter Instruction.

Note 2: To view or modify the servo parameters in CNC, please ensure the correctness of servo system connection and servo slave configuration.

3.1.1 System Parameters

Press SYSTEM —> PARAM —> CNC PARAM to enter into system parameter setting screen.

The system parameters can be set and modified on this screen. The current set parameters can be backed up, and system default parameter or backup parameters can be recovered.

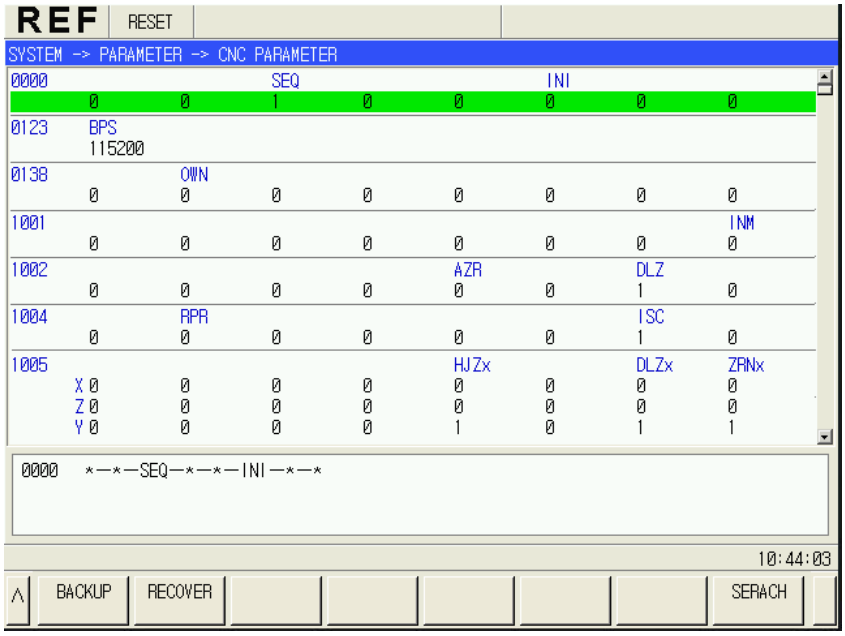
(1) Bit type parameters setting

Method 1:

① Select the parameter to be modified through keys , , , ; or press

SERACH softkey and input the parameter number, then press OK softkey, the cursor will move too the desired parameter.

② Press key INPUT to make the selected parameter modifiable. For example, the No.0000 parameter in the figure below:



③ Press the numeric keys to input 8 binary values, and then, press INPUT to complete the setting. (When the number input values is less than 8, fills the vacated bits with 0.)

④ Select other parameters through keys , , , .

Method 2:

① Select the parameter to be modified through keys , , , .

Select the bits to be modified through keys and .

③ Press INPUT repeatedly, to switch the parameter bit between 0 and 1.

④ Move the cursor to complete the setting.

⑤ Select other parameters through keys , , , .

(2) Numeric type parameter setting

① Select the parameter to be modified through keys , , , ; or press SERACH softkey and input the parameter number, then press OK softkey, the cursor will move to the desired parameter.

② Press key INPUT to make the selected parameter modifiable.

③ Input the numbers to be set through numeric keys, then press INPUT to complete the setting.

④ Select other parameters through keys , , , .

(3) Parameters backup and recovery

Before modification, the parameters can be backed up through BACKUP softkey. When the modification is erroneous or the parameter does not need to be modified, press RECOVER softkey, the backup parameters or system default parameters can be recovered.

➢ **Parameter backup**

① Press BACKUP on the parameter screen, BACKUP PARAM Are you sure that back up parameters? will be displayed.

② Press OK to back up the current set parameters.

➢ **Parameter recovery:**

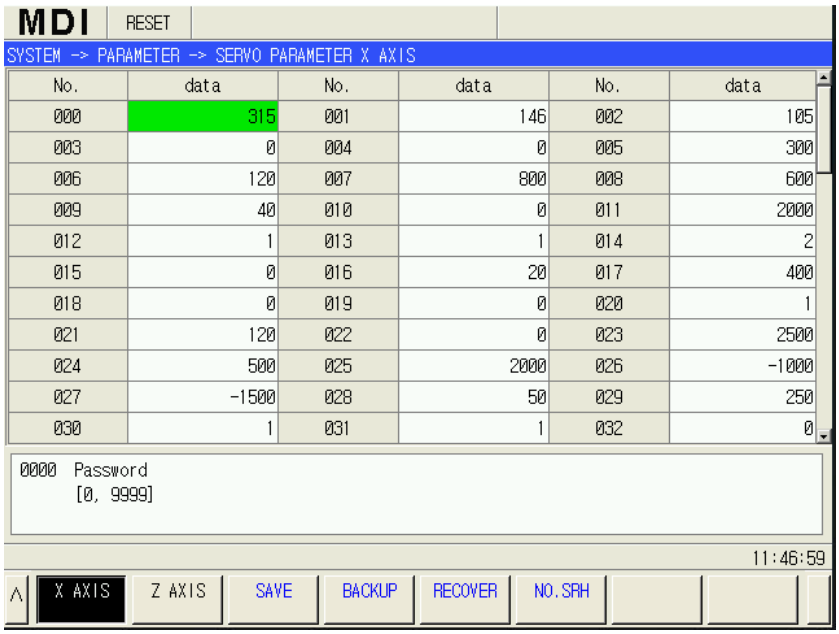
① Press RECOVER, RECOVER PARAM [USER] Recover backup para [DEFAULT] Recover default para [CANCEL] Cancel operation will be displayed.

② Press USER key to restore the backup parameters; Press DEFAULT to restore the system default parameters; press CANCEL to exit from the parameter screen.

3.1.2 Servo Parameters

(1) Modification and save

When the GSKLink communication is in normal state, on system screen, press PARAM —> SERVO PARAM, to enter into servo parameter screen.



Servo parameters can be viewed, modified, saved, backed up and restored through servo parameter screen on the CNC side.

Axes switching: Press X AXIS, Z AXIS or S AXIS to switch the displayed servo parameters.

Parameter modification: Press key INPUT and input the parameter value, or, input the parameter value directly, then press INPUT to complete the modification.

Parameter saving: after the modification, press SAVE to save the parameter. The saved parameter remains the same after servo is turned ON again.

Parameter backup: Press BACKUP, the following dialogue box will pop up, then press OK to back up the file.

Parameter recovery: Press RECOVER, the following dialogue box will pop up, the press OK to restore the backup file.

Select effective parameter: if the parameters are modified on the servo, after power-on, the system will issue prompt No.5030 "*the servo parameter in current parameter file of axis servo is inconsistent with the read servo parameter." Switch to the servo parameter screen this time, see Fig. 3-1-6 , then press SELCT EFF. PAR, see Fig. 3-1-7.

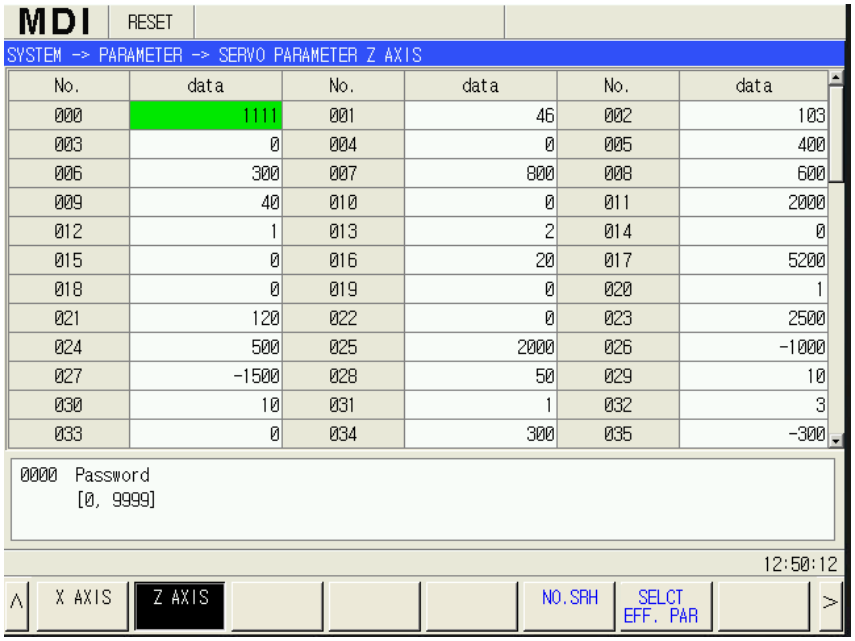


Fig.3-1-6

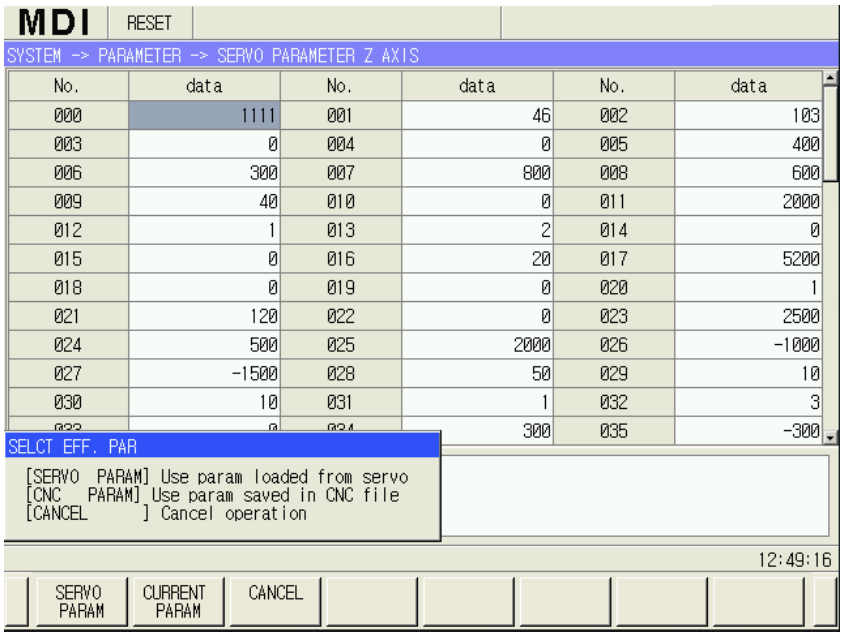


Fig. 3-1-7

Press CURRENT PARAM to validate the servo parameters in CNC; press SERVO PARAM to validate the parameters read from the servo; press CANCEL to return to the screen shown in Fig. 3-1-6.

(2) Restore motor default parameter

a. Refer to appendix B.1 to find the index value in the current software version of drive unit which is connected to the motor according to the motor type given on the motor nameplate.

b. Modify servo parameter PA1 to make it equal to the searched motor index value.

c. After modifying PA1, the system automatically update the default parameter corresponding to the motor. The parameter value is valid immediately after modification.

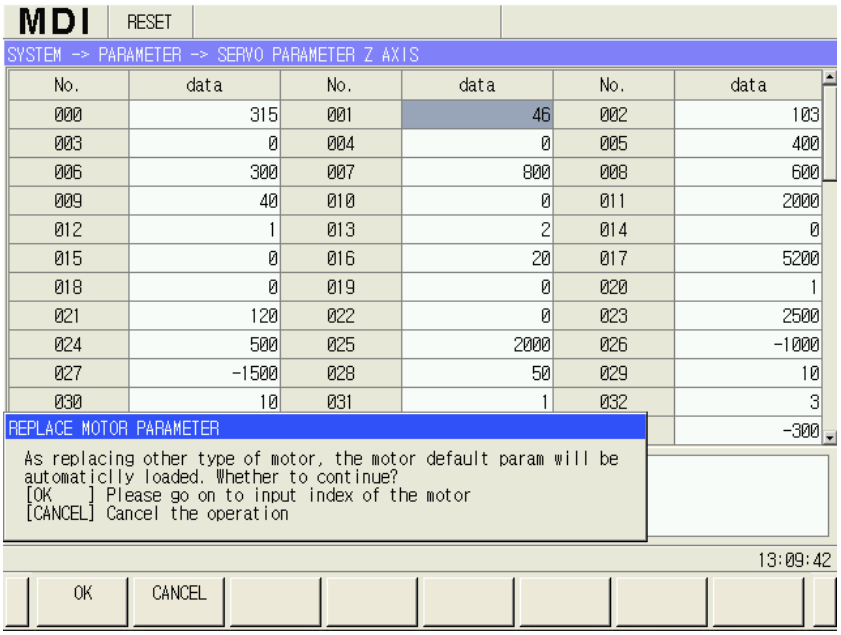
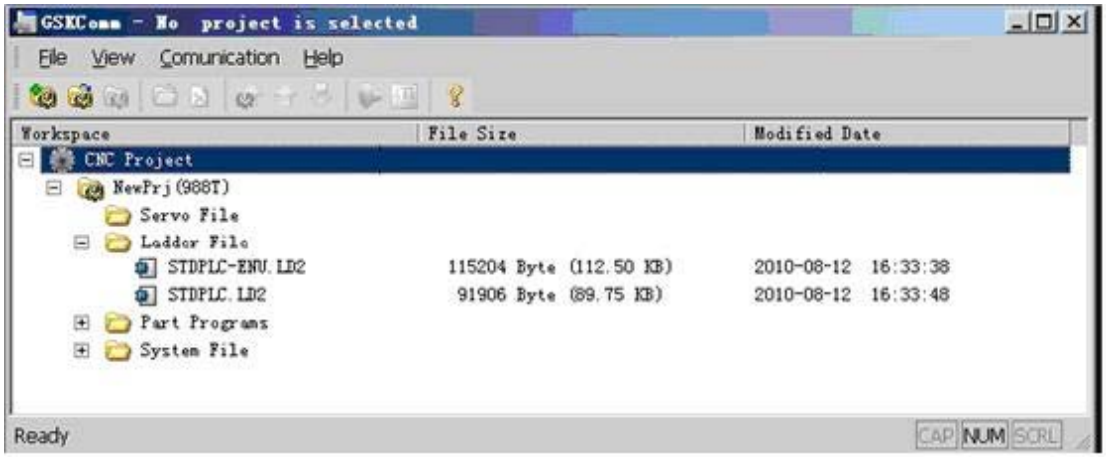


Fig. 3-1-8

3.2 Instruction of PC Communication Software GSKComm-M

This section is a simple instruction for the usage of the GSKComm-M during machine tool debugging. For the details, please refer to the *GSKComm-M Instructions* on the CD.

GSKComm-M is a communication management software especially provided for the machine tool builders. The GSKComm-M screen is shown as follows. It can realize the following functions: upload and download of files between PC and CNC, DNC communication, CNC parameter editing, part program management and editing, viewing tool compensation data and screw pitch error compensation data, ladder diagram editing, etc. It is convenient, efficient and reliable.



3.2.1 Preparation for GSKComm-M

(1) RS-232 series port connection

- **Connection between PC and CNC**

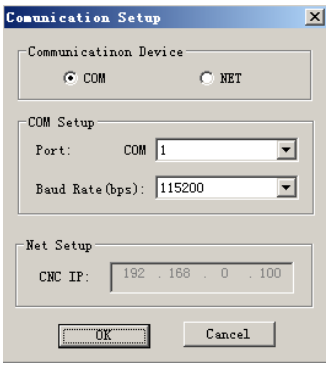
When both PC and CNC are power-off, the communication cable should be connected as follows: DB9 male is plugged into the RS-232 communication interface on the CNC; DB9 female is plugged into the 9 pins serial ports on the PC (COM0 or COM1).

- **Baudrate setting in CNC**

The baudrate is set by parameter No.0123. When data transmission is processed between CNC and PC, the setting value should not be less than 4800. (ex-factory value:115200)

- **Baudrate setting in PC**

After the communication software is executed, left-click the menu and select “Communication－>Communication Setup”, shown in right figure.



Setting: select the serial port communication.

Port selection: select ports used for communication (COM1, COM2, COM3, COM4)

Baudrate: Select the baudrate (4800, 9600, 19200, 18400, 57600, 115200) (unit: bps)

(2) Network connection

- **Connection between PC and CNC:**

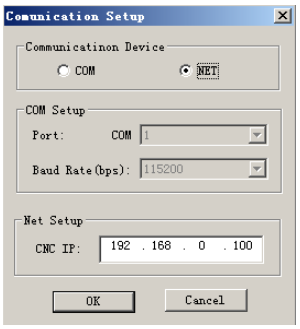
Connect the network port of GSK988T to the PC or router with normal network cable.

- **IP setting on CNC:**

Press SETTING —> CNC SETTING —> ETHENET, to enter into IP setting page to set the IP address and gateway.

- **IP setting on PC:**

After the communication software runs, left-click the menu, and select “Communication—>Communication Setup”, shown in right figure.



Communication setting: Select network communication.

Network setting: Fill in the IP set in CNC.

(3) Authority setting

During upload and download using GSKComm, corresponding authority should be set in advance, otherwise, the operation will fail.

Data to be downloaded	CNC least authority level	Remark
PLC files	2 level	
parameters	3 level	Parameter switch is ON
Part programs	3 level	Program switch is ON
Macro variables	4 level	Program switch is ON
Tool compensation data	4 level	
Pitch error compensation data	5 level	Parameter switch is ON
Tool life files	5 level	

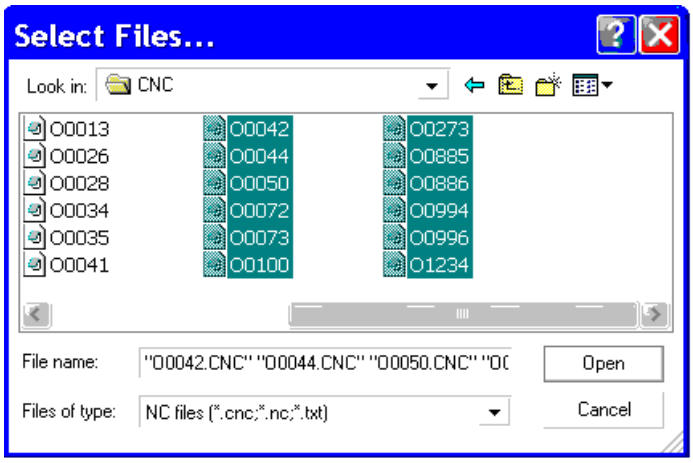
3.2.2 File Download (PC→CNC)

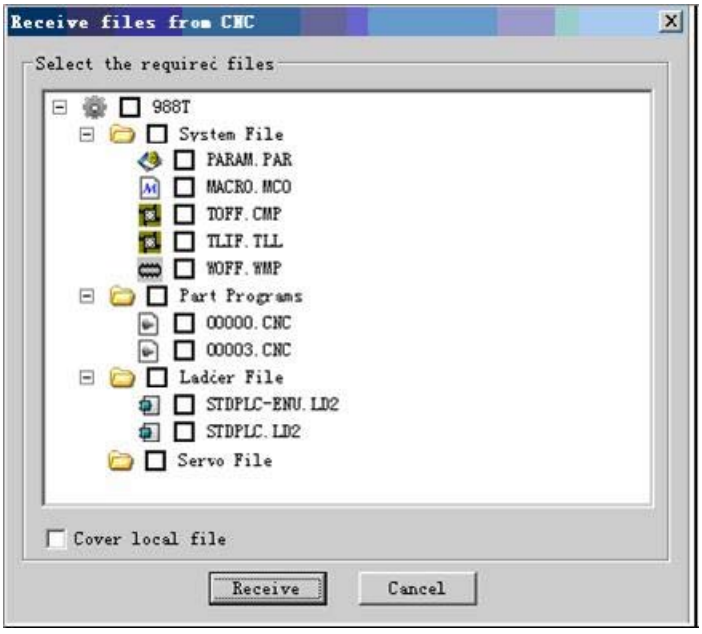
Through GSKComm, files in the PC can be transferred to CNC altogether or one by one.

(1) Add files

First, press the type of file to be added (for example, system file, part program file or ladder diagram file)

Then, press  or right-click, select “Add Files”, a dialog box for adding file will pop up (shown in the left figure), select the desired file (hold down “shift” key to select more files), then click “Open” to complete the action.





(2) Add multiple files

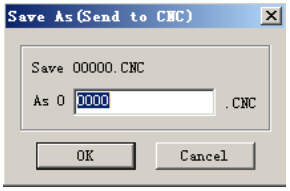
First, select the project to be transferred; then, click  or right-click the project and select “Send to CNC”, the following dialog box will pop up. (Shown in the right figure above)

In this dialog box, click the left options to select the files to be transferred. Arrow “—>” points to the file name saved in CNC, double-click it, you can change the file name.

Click “Start sending” you can transfer the selected file (with the saved file name) into CNC.

(3) Download single file

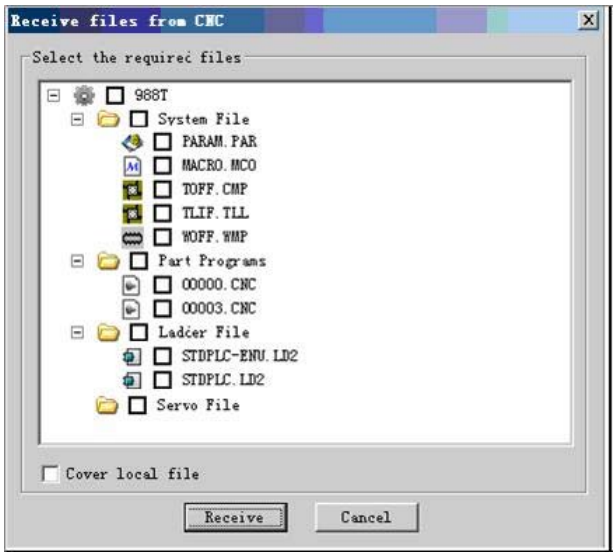
Select the file to be downloaded, then click  , or right-click the file and select “Send to CNC”, a dialog box will pop up. You can change the file name to be saved in the CNC, and then click “OK” to transfer the file.

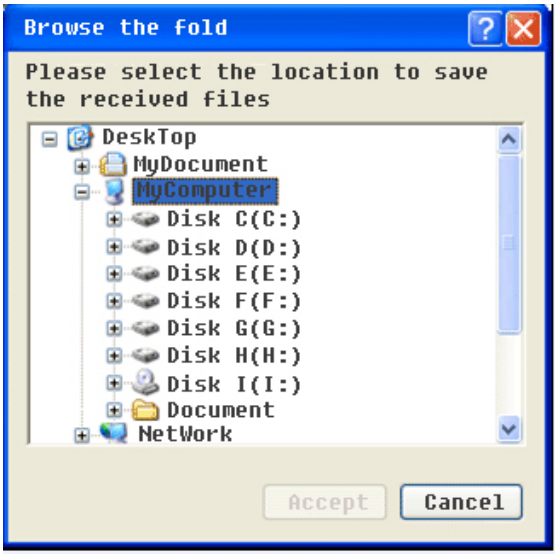


3.2.3 Upload File (CNC→PC)

First, select a project, then, click  , or select menu "Communication—>Receive Files from CNC”, a dialog box will pop up, (shown in the left figure below). Select the file to be uploaded, then

click “Receive”, a “Browse File” dialog box pops up (shown in the right figure below).





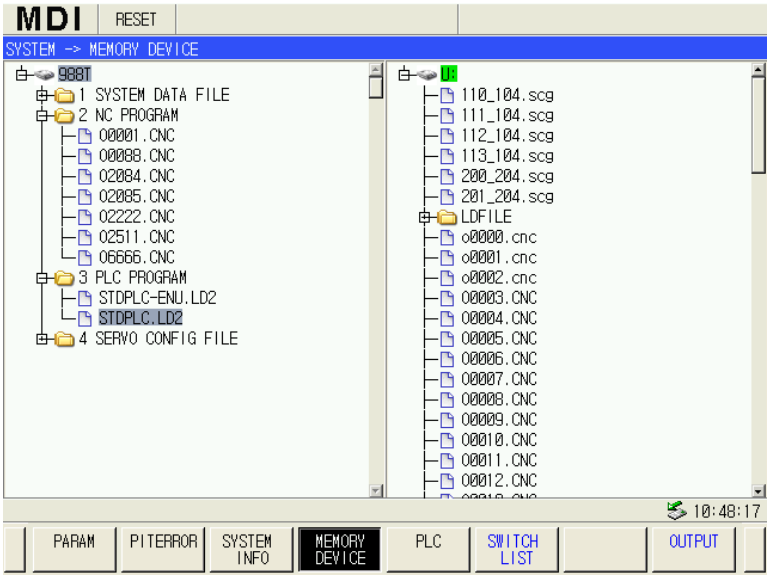
Select the file folder in which the uploaded file to be saved, and click “OK” to upload the selected file.

3.3 Usage of U Disk

The U disk function in GSK988T supports the bi-directional transmission of files involving machining program, PLC program, parameters, tool compensation data and pitch error compensation data. It can be operated on three screens: file management screen, program screen and ladder diagram screen.

3.3.1 File Management Screen

When U disk is already inserted in the USB port, press SYSTEM → MEMORY DEVICE to enter into file management screen.



Under this screen, bi-directional transfer of system files (system parameters, tool compensation data, pitch error compensation data etc.), ladder diagrams and part programs can be executed. The procedure is shown as follows:

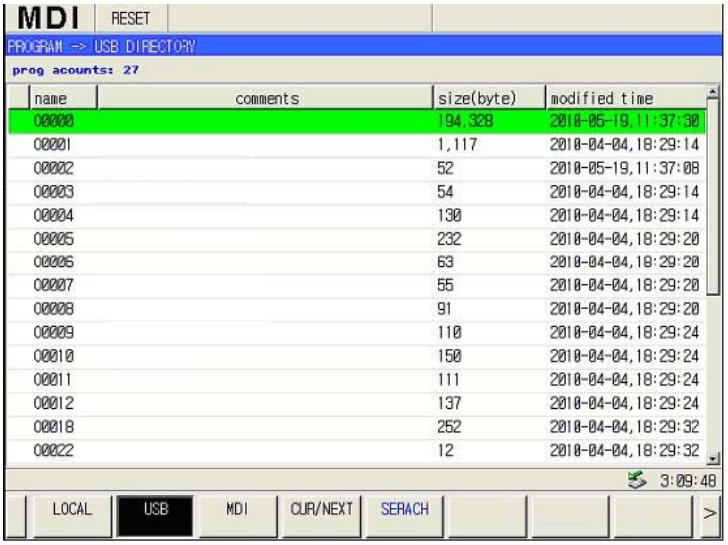
Press SWITCH LIST to switch between the system content and U disk content.

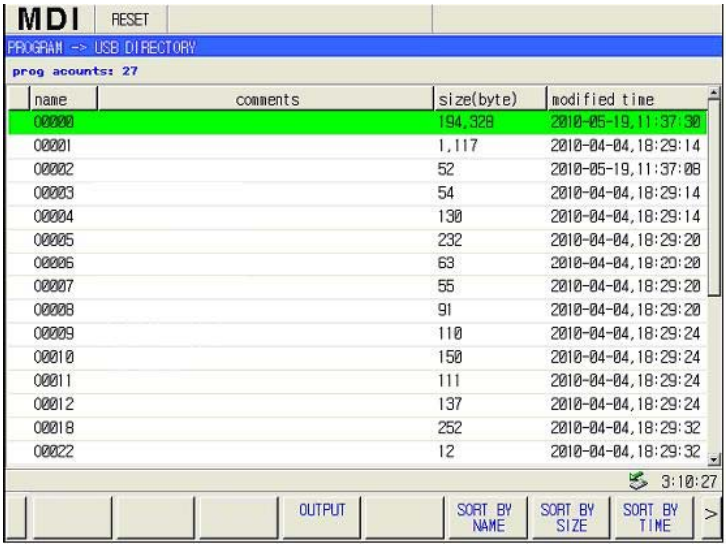
Press OUTPUT softkey to copy the selected program into local directory or U disk directory.

3.3.2 Program Screen

> The operation of U disk directory is the same as in the local directory. In this section, we only introduce the program transfer in U disk. For details, please refer to GSK*988T User Manual.*

When the system USB port is inserted with U disk, press PROGRAM to enter into program directory (left figure below), press >, the extended softkeys will be displayed. Then, press USB to enter into U disk directory screen, operations to the programs in U disk directory such as load, open, copy, paste, create, save as, delete, rename, search, etc, can be performed.

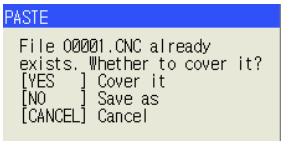




➢ **Program bi-directional transmission**

① Press LOCAL and USB to switch between the local directory and U disk directory.

② Move the cursor to the program to be copied through ⇧ and ⇩



Then, press > to view the extended softkeys (the right figure above), Press OUTPUT to copy the selected program into local directory or U disk directory.

③ When the copied program already exists, a dialog box pops up (see the right figure). Press "Yes" softkey to cover the existed program; or press "No", a dialog box 

pops up, then input the program name for saving; press "Cancel" to cancel the operation.

Note 1: When transmission is made from the U disk directory to local directory, the machining programs can be read only when it is stored in the root directory "NCPROG" file in the U disk.

Note 2: When transmission is made from local directory to the U disk directory, if the "NCPROG" file does not exist in the U disk, the file will be created automatically, and the machining programs will be output to the files.

3.3.3 PLC Screen

When the USB port is inserted a U disk, press PROGRAM on the ladder diagram screen, the screen is shown in Fig. 3-3-4 (Local directory screen). Press USB to switch to the U disk directory screen, shown in Fig. 3-3-5.

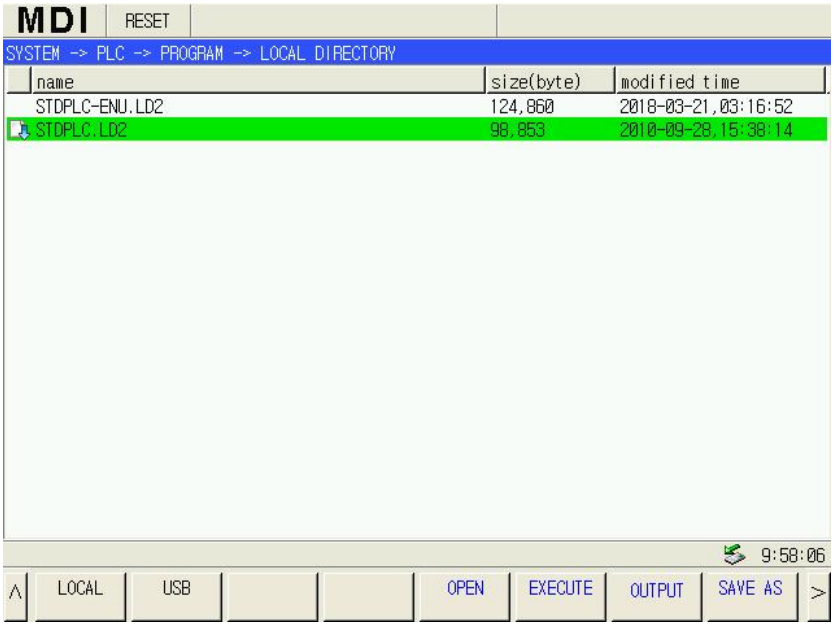


Fig. 3-3-4

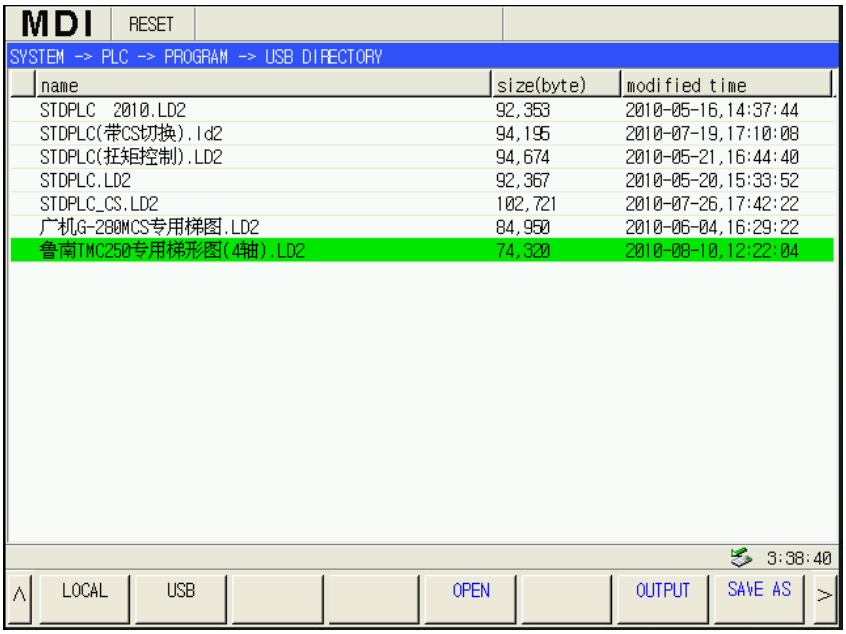


Fig. 3-3-5

Programs in the U disk directory can be copied to local directory through softkey OUTPUT, vice versa.

Take the U disk for example, the procedures are shown as follows:

① Press softkey USB to enter into U disk directory;

② Select the ladder diagram programs to be copied through ⇧ and ⇩, then, press OUTPUT to copy it to the local directory.

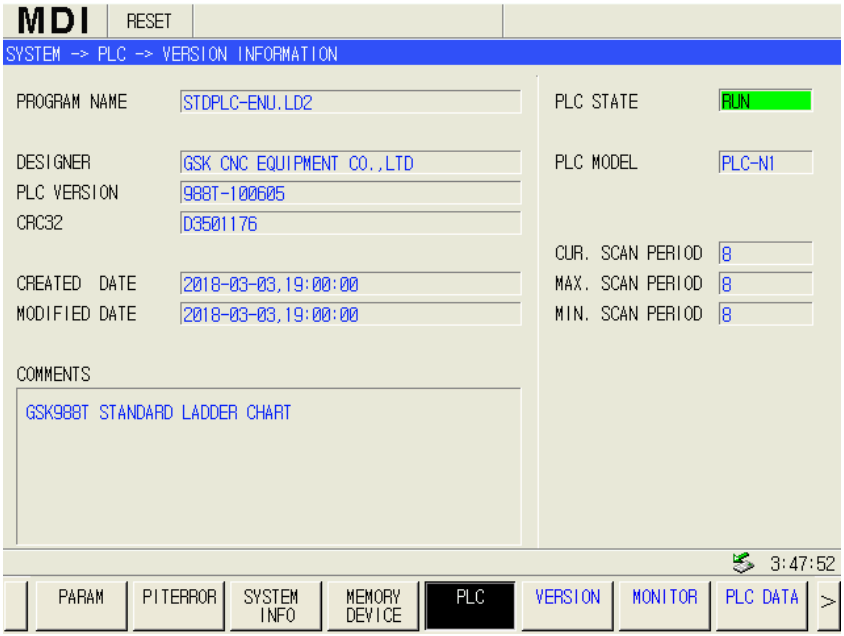
Note 1: When PLC transmission is made from the U disk directory to local directory, the PLC programs can be read only when it is stored in the root directory "LDFILE" file in the U disk.

Note 2: When PLC transmission is made from local directory to the U disk directory, if the "LDFILE" file does not exist in the U disk, the file will be created automatically, and the PLC programs will be output to the files.

3.4 PLC Operation

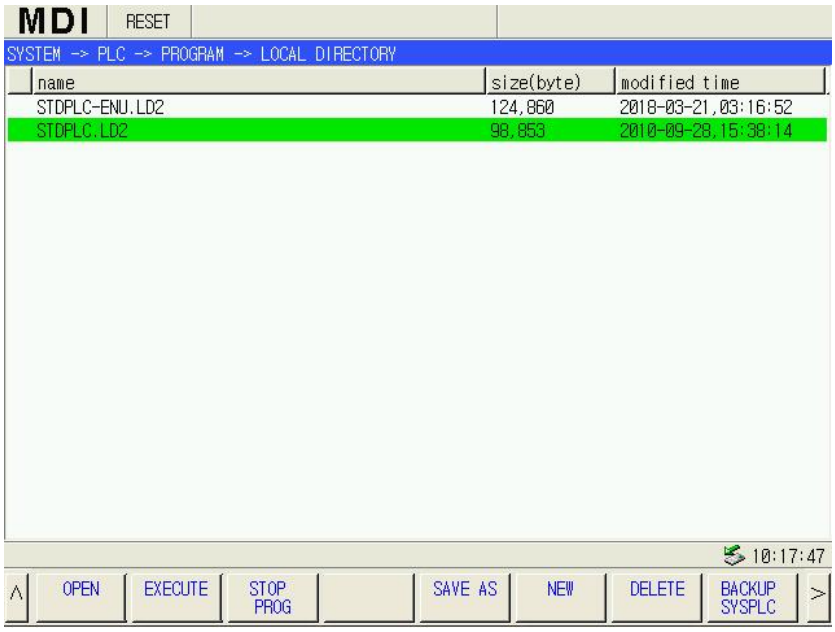
Press function key SYSTEM and then press softkey PLC to enter into PLC screen. This screen includes pages such as version information, monitor, PLC data, PLC state, program directory. Press corresponding softkeys, you can view the desired content.

After entering the PLC screen, the contents of VERSION is displayed. The version page includes the information about the PLC version, current running PLC program and the running state, etc.



3.4.1 PLC Execution and Stop

On PLC screen, press softkey [>], then press [PROGRAM]→[LOCAL], the following screen is displayed:



On this page, you can select PLC program through [⇧] and [⇩], then operations such as edit, running, stop, save, create, delete and backup can be performed.

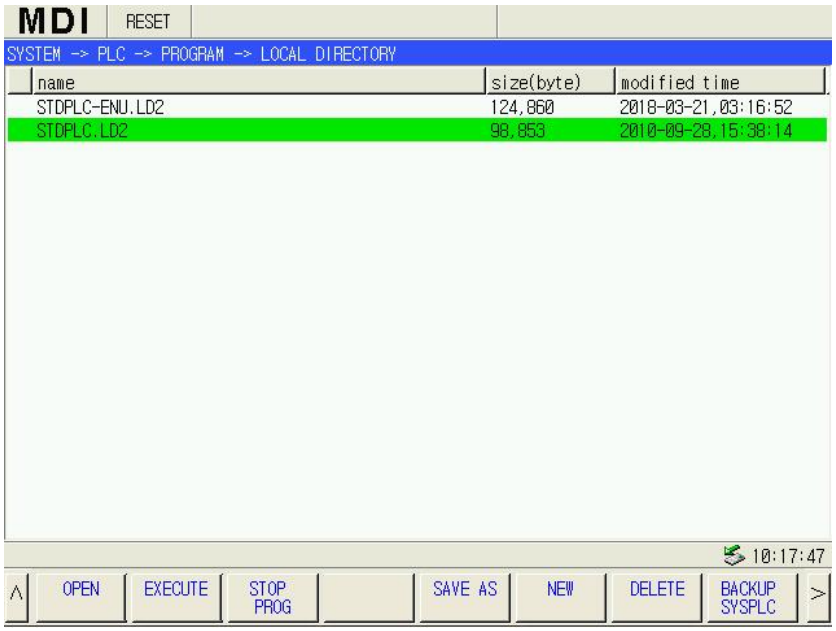
➢ **Execution of PLC programs**

Select the PLC program through [⇧] and [⇩], then press [EXECUTE] to start running.

Note: The current running PLC program is marked with .

➢ **Stop PLC program execution**

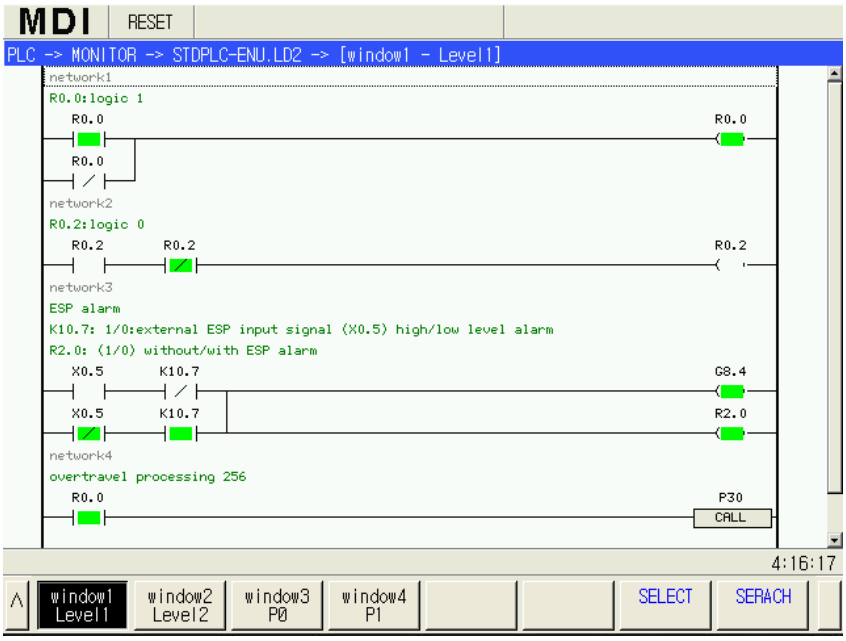
Move the cursor to the current running program through ⇧ and ⇩, then press STOP PROG, the system will be in no PLC running state.



3.4.2 PLC Monitoring and Diagnosis

(1) Monitor the PLC program state

On the PLC screen, press softkey MONITOR to enter to the monitoring display screen for the current running PLC program.



You can view the state of current contact, coil conducting ON/OFF and the current value of timer and counter. When the contact and coil conduction is ON, it is indicated by green color; if not, the

color is the background color of the screen. For example: X0.5 ⊣■⊢ means the contact X0.5 is conducted, Y25.2 ⊣( )⊢ means the coil Y25.2 is not conducted.

➢ **View blocks**

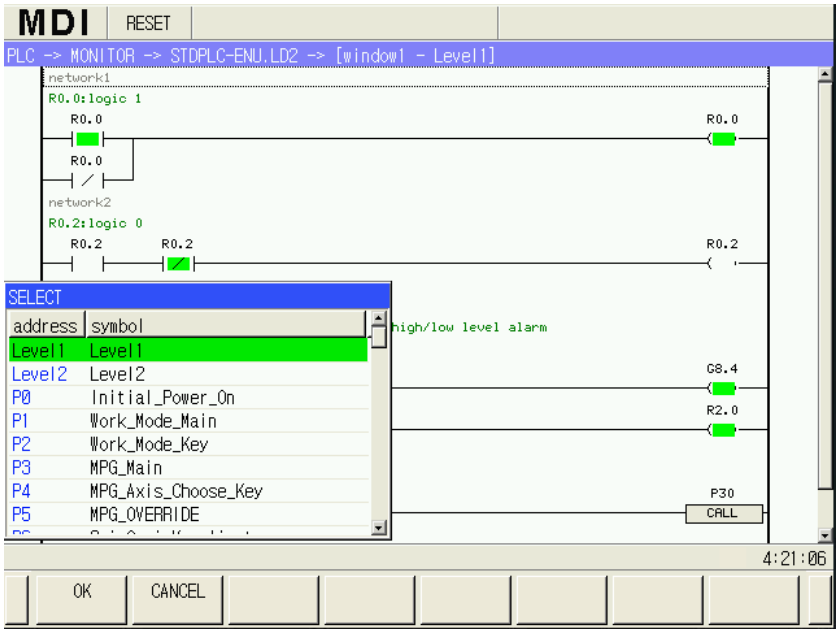
On monitoring page, there are four softkeys for monitoring four blocks: window1 Level1, window2 Level2, window3 P0, window4 P1. Each of them corresponds to a block and the corresponding PLC will be displayed on the screen.

> **Note 1: Softkeys for windows 1~4 are shortcut keys which enable quick view of the corresponding blocks.**
> **Note 2: The blocks corresponding to windows 1~4 can be changed, but the change will not be effective after power-off. The default block after power-on is the first four blocks in the PLC programs.**

➢ **Select block**

① Select the screen as needed.

② Press softkey SELECT, the following figure is displayed:



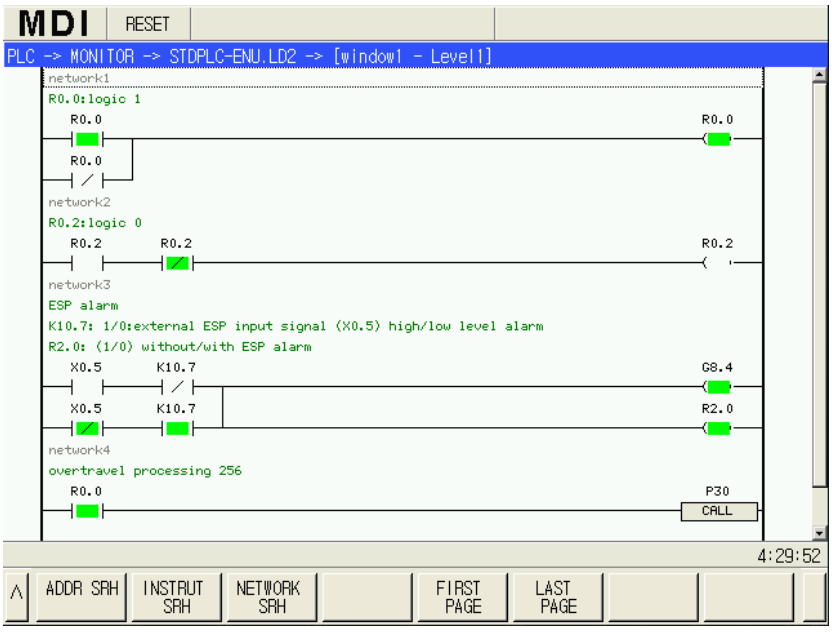
③ Press keys , , , , to select the desired window.

④ Press softkey OK to complete the selection, then, return to the previous menu; press softkey CANCEL to cancel the selection and return to the previous menu.

➢ **Search for parameters, commands and network**

Select the window in which the command, parameter or network to be searched, i.e. press window1 Level1, window2 Level2, window3 P0, window4 P1 to display the corresponding blocks of PLC program, then, search for the command, parameter or network.

② Press softkey [SERACH] to enter to search page, shown as follows:

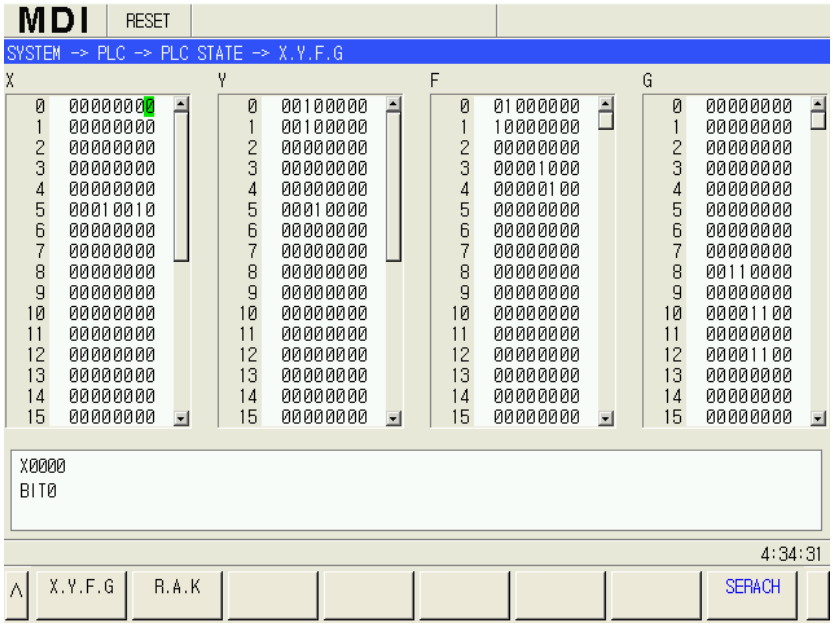


③ Press softkeys [ADDR SRH], [INSTRUT SRH], [NETWORK SRH] respectively you can search for the parameters, command, network on the corresponding screen, and move the cursor to the corresponding position.

④ Press [FIRST PAGE], [LAST PAGE] to move the cursor to the first line and last line of the block.

(2) PLC I/O state diagnosis

On PLC screen, press [>] and [PLC STATE] to enter to PLC state display page, as shown in the following figure.



➢ **View the state of signals**

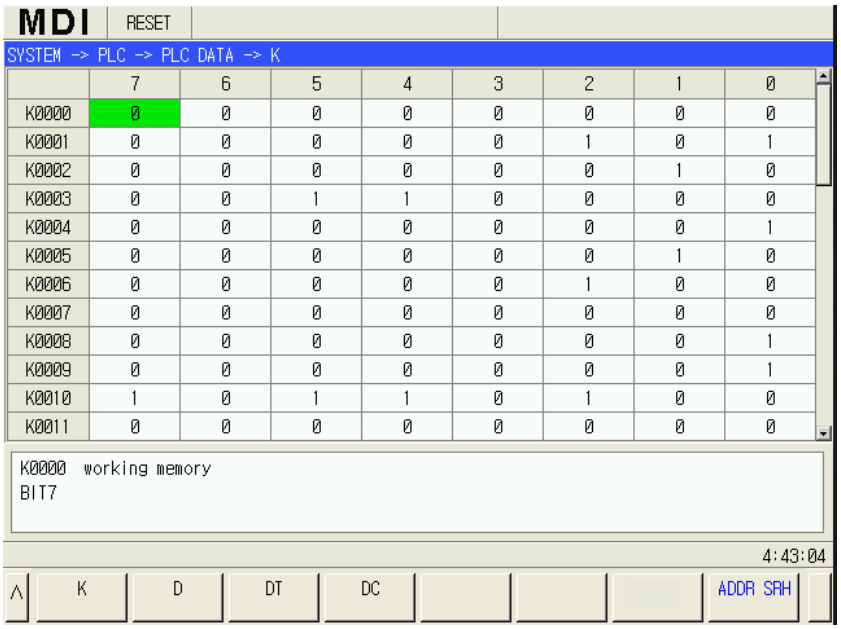
Press softkey [X.Y.F.G], the state of signals X, Y, F, G will be displayed on the screen; press softkey [R.A.K], the state of signals R, A, K will be displayed.

Press ⇦ or ⇨ to switch between softkeys X, Y, F, G signal and R, A, K signal.

Press , , ⇧, ⇩ to view the information about X, Y, F,G signals or R, A, K signals.

3.4.3 PLC Data Viewing and Setting

On PLC screen, press PLC DATA to enter into PLC data state page. It includes the setting of K, D, DT, DC parameters.



(1) K parameter setting

On PLC data page, press K softkey to enter into parameter K setting page, shown in the figure above:

Parameter setting method:

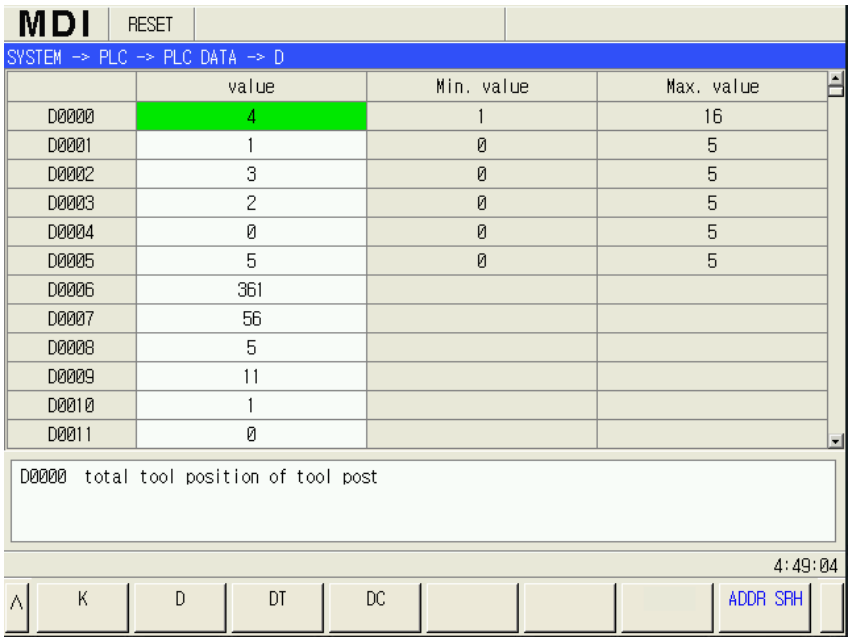
① Press keys , , ⇧, ⇩, ⇦, ⇨, you can select the parameter statues bit to be modified; or press softkey ADDR SRH to input the K variable to be selected, then press OK and move the cursor to the parameter. The meaning of the status bit is displayed at the bottom of the screen.

② Press INPUT repeatedly in K variable status bit to switch between 0 and 1, modify the status of the selected K parameter status bit.

③ Press ⇧, ⇩, ⇦, ⇨ to move the cursor to complete the modification.

(2) D parameter setting

On PLC data page, press D to enter to the D parameter setting display page, shown in the following figure.



Parameter setting method:

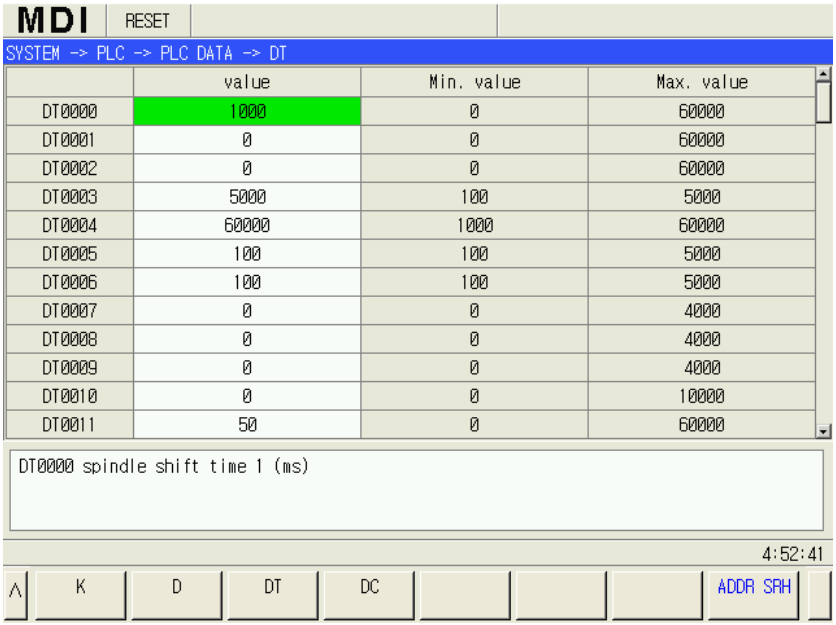
① Press keys , , , , , , you can select the D parameter to be modified; or press softkey ADDR SRH to input the D parameter to be selected, then press OK and move the cursor to the parameter. The meaning of the status bit is displayed at the bottom of the screen.

② Press INPUT to enable the selected D parameter to be modifiable.

③ Input the modified value, and press INPUT key again to finalize the modification.

(3) DT parameter setting

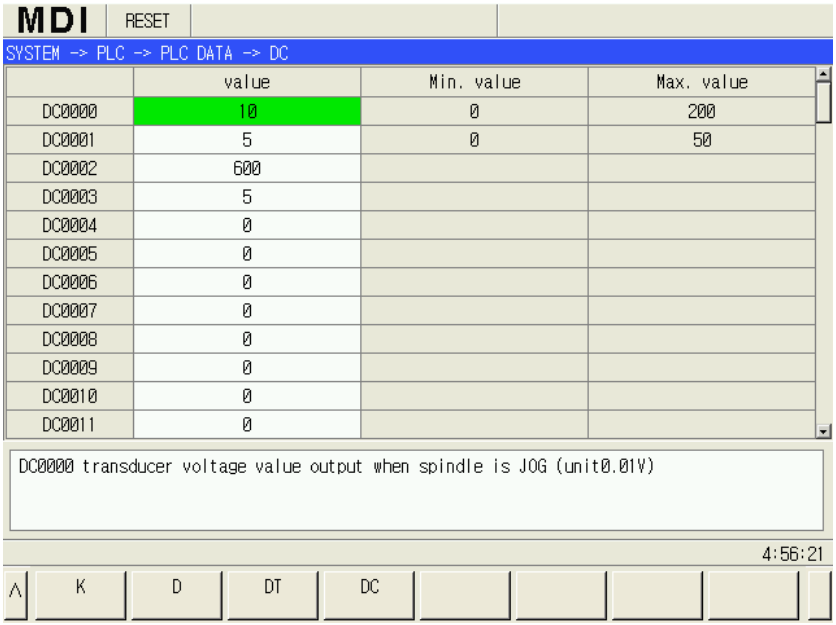
On PLC data page, press DT to enter to the DT parameter setting display page, shown in the following figure.



Parameter setting method: the same as D parameter setting

(4) DC parameter setting

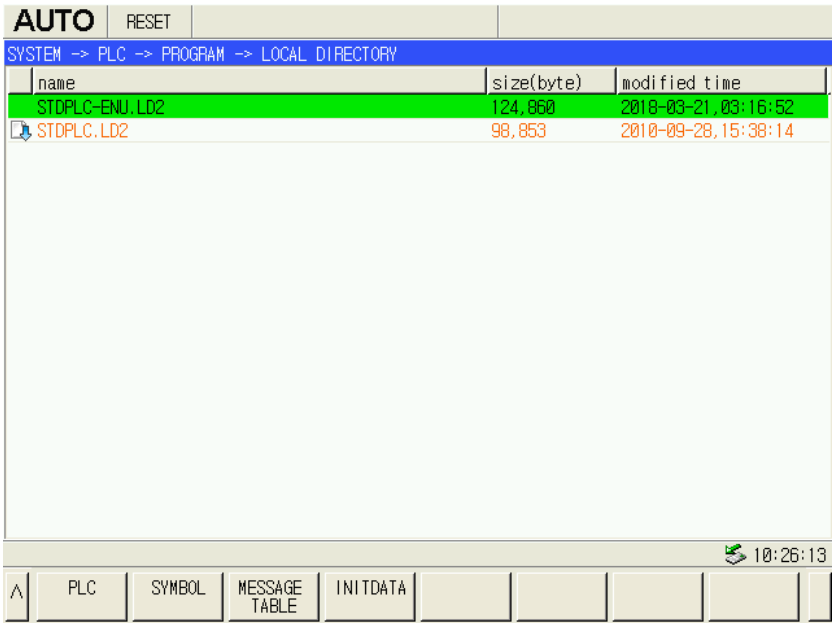
On PLC data page, press DC to enter to the DT parameter setting display page, shown in the following figure.



Parameter setting method: the same as D parameter setting

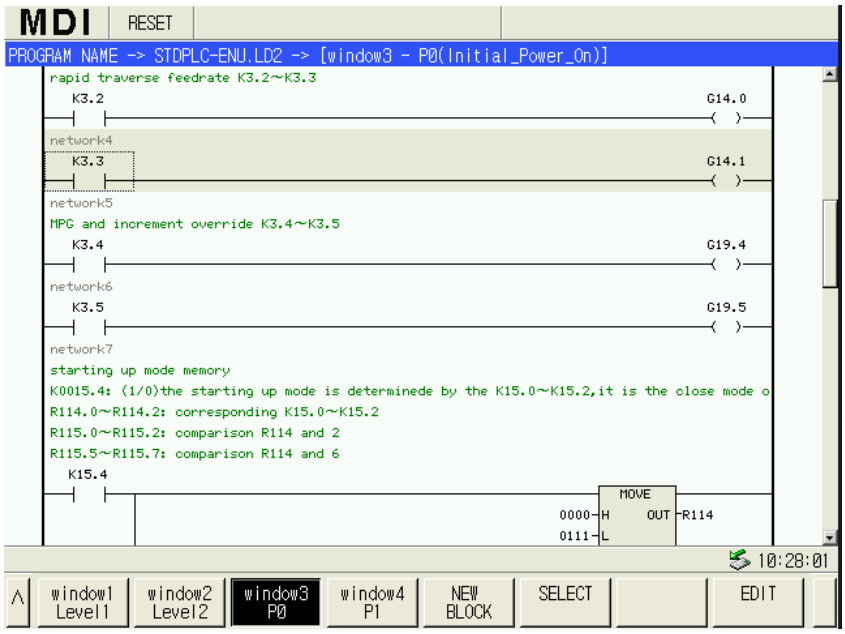
3.4.4 PLC On-line Modification

On PLC screen, press > → PROGRAM → LOCAL to enter into PLC program directory page, press ⇧ and ⇩ to select the program to be edit, then press softkey OPEN to enter into edit page. You can edit the ladder diagram, symbol table, information display table, initialized data table.



(1) View and edit PLC

On the page shown in the above figure, press PLC to enter into PLC display and editing pages. Shown as follows:



In this figure, the position where the cursor locates is indicated with dashed frame, and the background color of the network area is darker.

Press window1 Level1, window2 Level2, window3 P0, window4 P1 respectively, the corresponding blocks will be displayed and the block name will be displayed on the upper area of the window.

① **Select blocks to the windows**

Press window1 Level1, window2 Level2, window3 P0, window4 P1 according the block to be modified, and then, press SELECT you can select a block to be displayed on the corresponding window.

Press , , , to select the block, then press OK to complete the action and return. At this time, the address of corresponding block is shown on the screen. For example, window1 Level1 indicates that the window 1 corresponds to the Level 1 block, when window1 Level1 is pressed, the content of Level 1 block is displayed on the screen.

② **Create a new block**

Press window1 Level1, window2 Level2, window3 P0 or window4 P1 to select a window on which a block is needed to be created, then, press NEW BLOCK and enter the block name, press OK to complete the action.

③ **Edit program**

Select a window to be modified, then press EDIT to enter into edit program page (see Fig. 3-4-14), press > to display the extended keys.

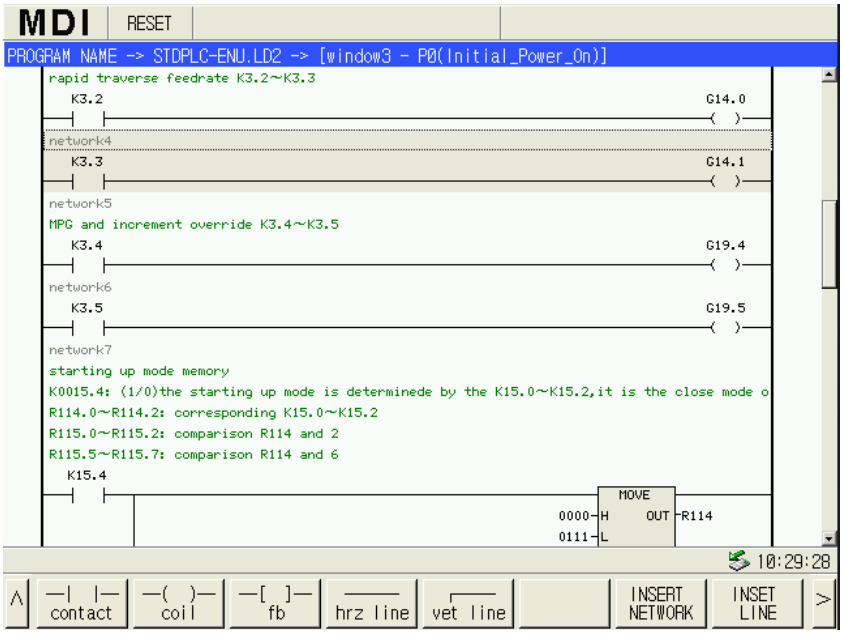


Fig. 3-4-14

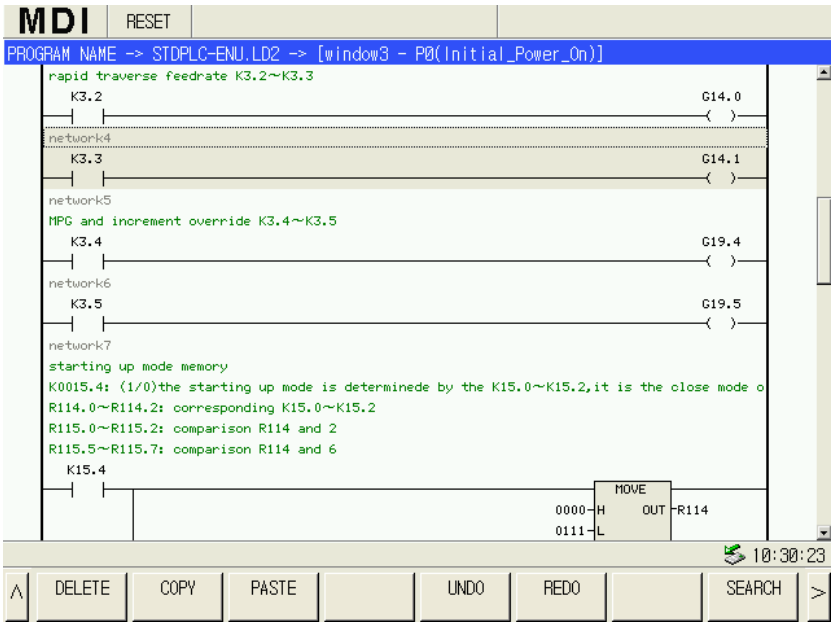


Fig. 3-4-15

A. Press  ,  keys to move the cursor to the position to be edited.

B. Press INSERT NETWORK to insert a network in front of the network where the cursor locates.

C. Press INSET LINE to insert a line behind the line where the cursor locates.

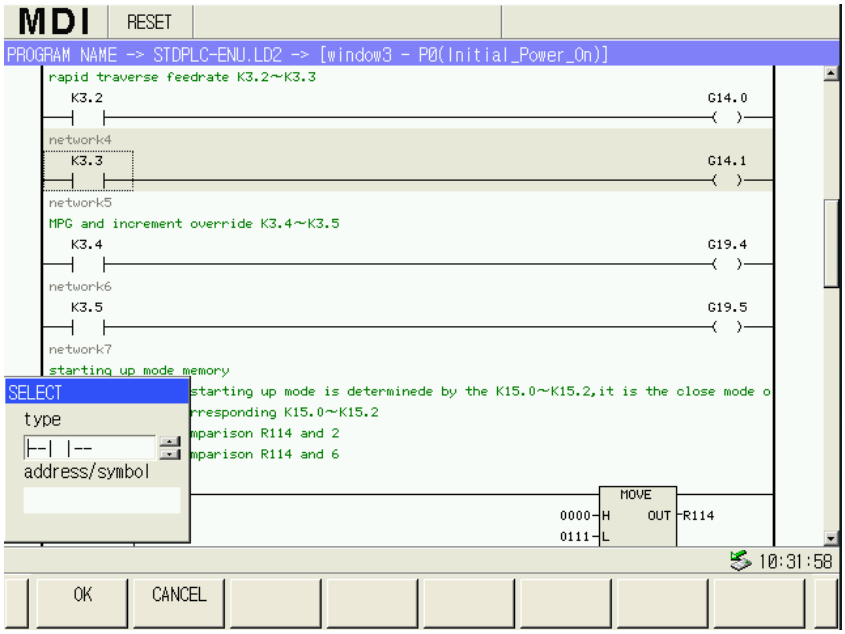
For example:

● Press —| |— contact, the following figure is displayed.

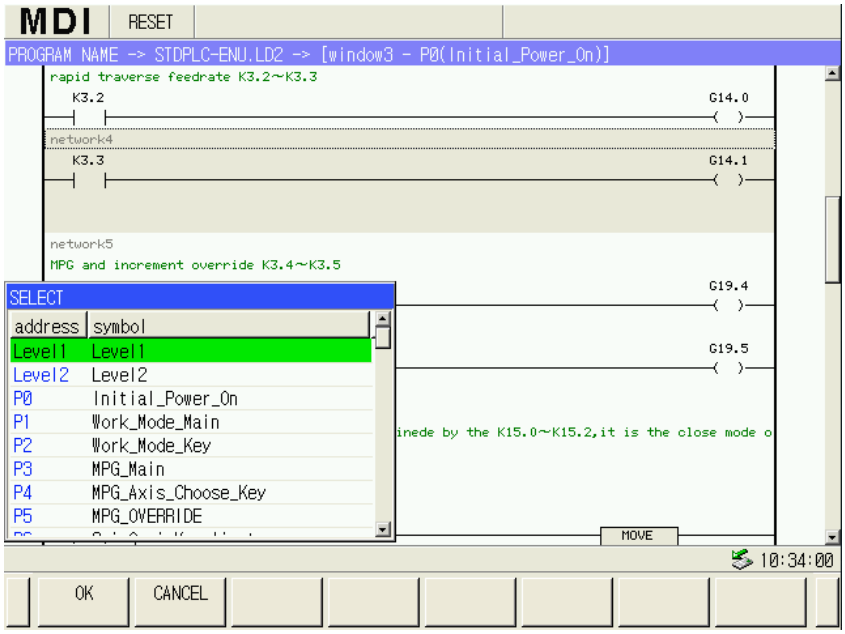
Move the cursor to the “Type” selection box at this time, then, press  to switch between the normally-open contact and normally-closed contact.

Press CHANGE to switch to the “Add/Symb” edit box, and enter the address/symbol, then press OK softkey or “Input” key to complete the action.

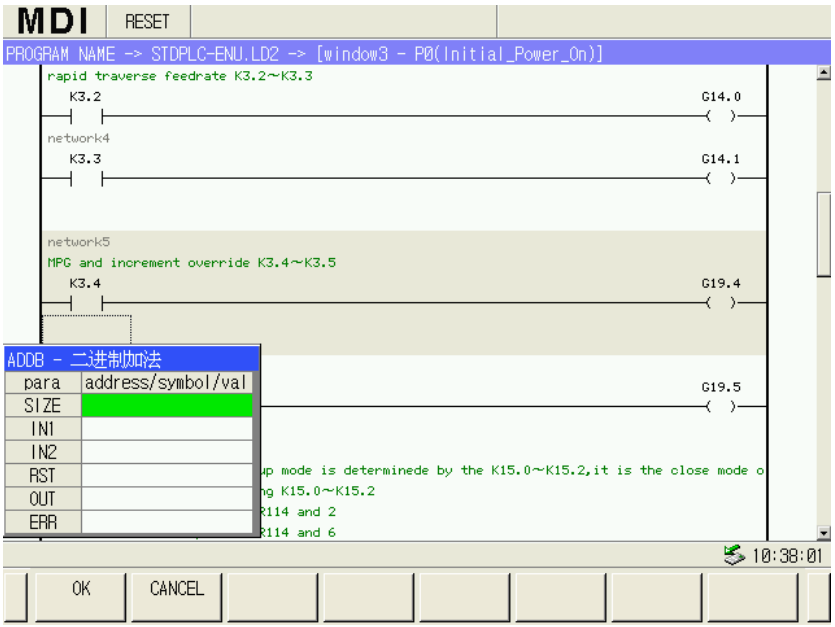
- Press softkey —( )— coil , the following operation is the same as —| |— contact softkey.



- Press softkey —[ ]— fb, the following figure is displayed:



Press ⇧, ⇩ to select the function command to be inserted, for example, ADDB in the figure above, press OK softkey or press INPUT, the following figure is displayed:



Press ⇧ or ⇩ in the edit box to select the parameter to be edited, then enter address or data, and press INPUT to confirm the modification, after all the editing is done, press softkey OK to complete the action.

● The operation of other function commands is the same as described above.

You can delete, copy, paste and edit all the components, lines and network at this time. You can cancel the last step or re-do the step.

● Press softkey SERACH to switch to the search page, then you can press ADDR SRH, INSTRUT SRH, or NETWORK SRH and enter relevant parameters, command, or network. Move the cursor to the position where the searched parameter (or command, network) locates, then press FIRST PAGE, LAST PAGE or move the cursor to the head of a block or end of a block.

● In a similar way, press DELETE to delete component, line or network.

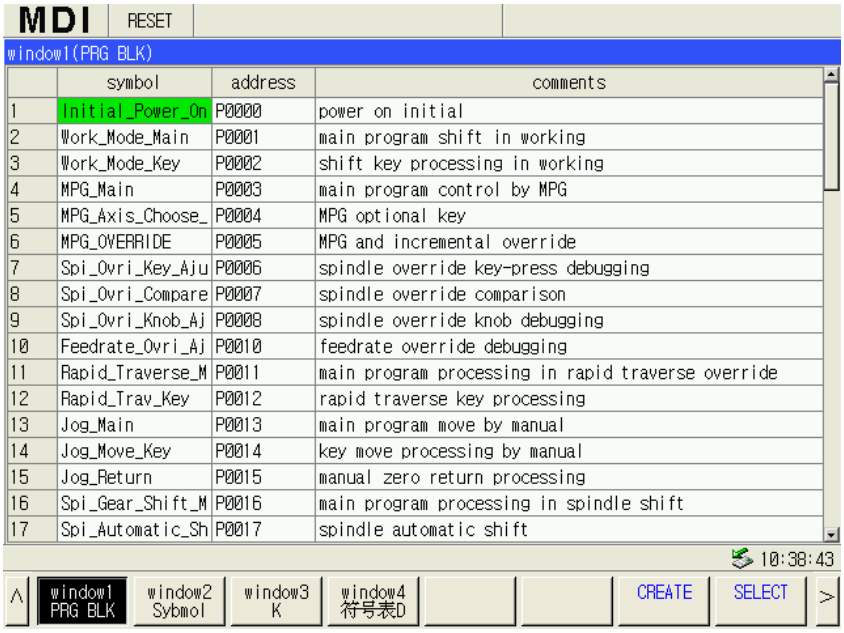
● In a similar way, press COPY to copy component, line or network. After the copy is complete, press key ⇧, ⇩ to move the cursor to the desired network, line or component , then press PASTE.

● Press UNDO to undo last operation. You can cancel up to 20 steps of operation.

● Press REDO to redo the cancellation.

(2) View and set symbol table

On PLC edit page, press SYMBOL to enter into symbol table display page:



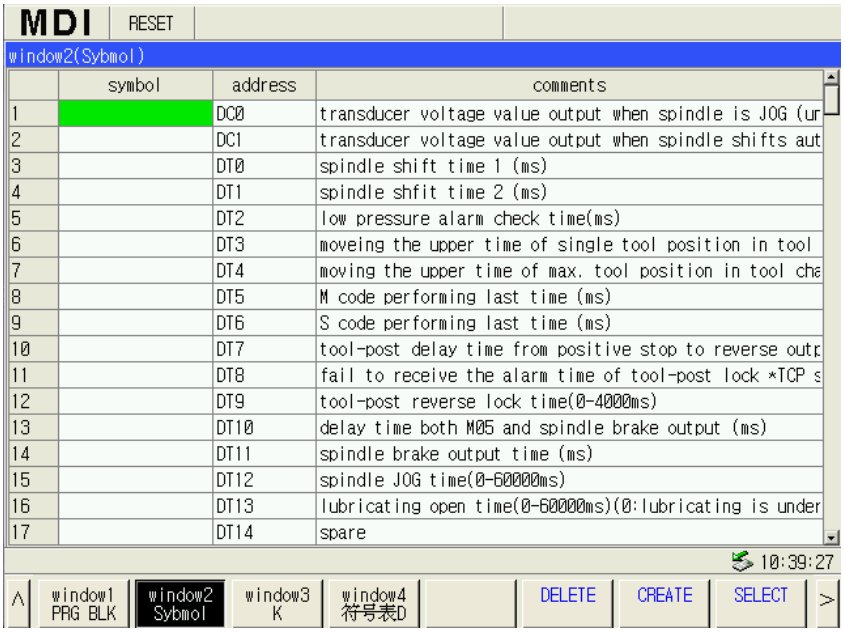
① Press softkeys window1 PRG BLK, window2 Sybmol, window3 K, window4 符号表D respectively, you can select the symbols to be displayed on the window. The window name and corresponding symbols table name is shown on the upper area of the screen.

Press softkey SELECT to select a symbol table for each window.

Press CREATE to create a new symbol table and it is displayed on the current window (Note: if there is an empty window, the created symbol table will be displayed on the empty window preferentially.

Press window1 PRG BLK, the block symbol table is displayed and the corresponding address table is displayed as well.

② Press window2 Sybmol, the symbol table is displayed, shown as follows:



Notes for parameter addresses X, Y, DC, DT, T, R are displayed in the symbol table.

Press , , , , , , to select and view all parameter addresses.

Press softkey DELETE to delete the selected symbol table.

Press > to show the extended softkeys.

Press SERACH and enter the parameter address to be searched, locate the cursor to the address.

Press INSERT LINE to insert a null line below the line where the cursor locates.

Press DELETE LINE to delete the line where the cursor locates.

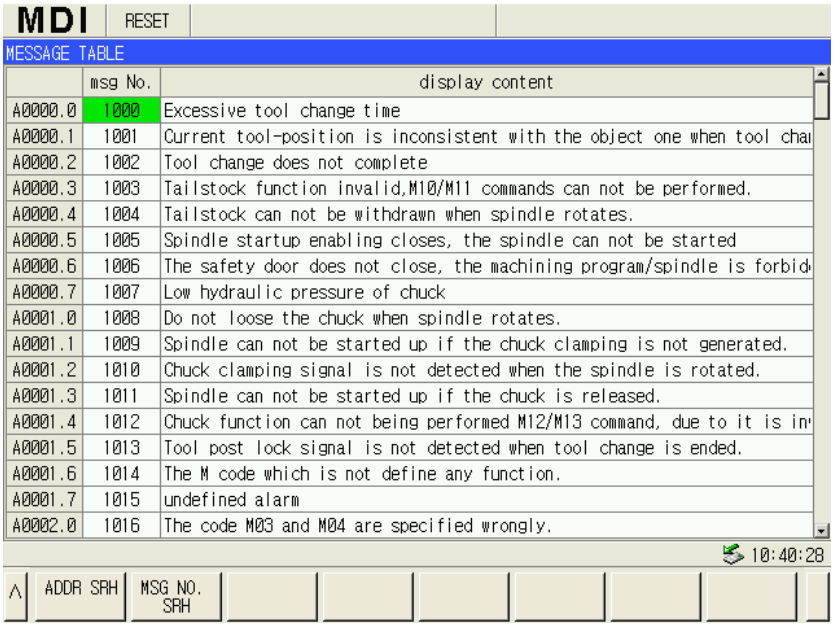
③ The operation after press window3 K, window4 符号表D is the same as window2 Sybmol.

④ Modify and edit the symbol table (the block symbol table cannot be modified here)

Select the symbol table to be modified, then press , , , , , , to select the symbol (or address, annotation) to be modified, press INPUT and input symbol, address or annotation, then press INPUT again to complete the modification.

(3) View and Modify the message table

On edit page, press MESSAGE TABLE to enter to message table, shown as follows:



In this information display table, PLC alarm information address A, corresponding information number and content are shown. Press , , , , , , to check these information.

① **Modification of information number and content:**

Press , , , , , , to select the information number or content to be modified, then press INPUT and input the desired information number or content, press INPUT again

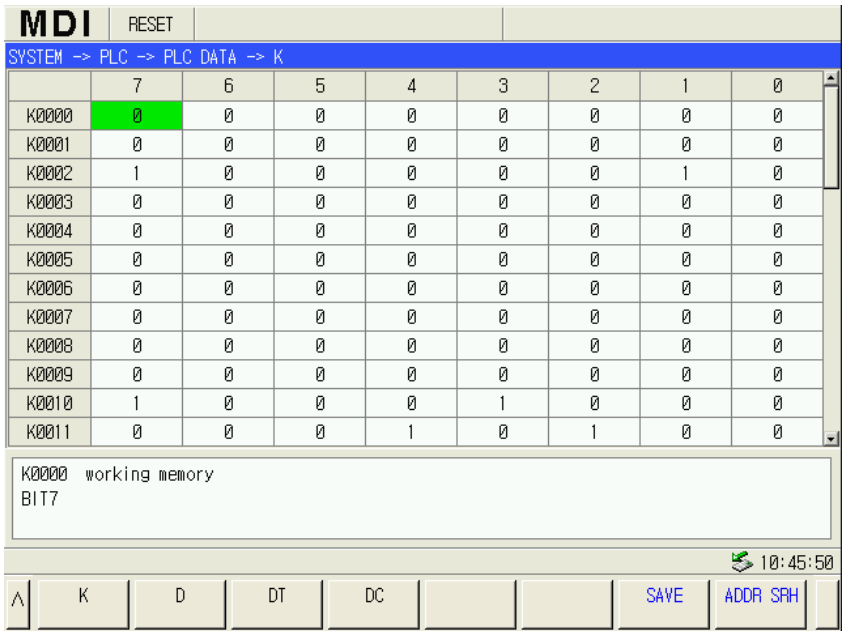
to complete the modification.

② **Search for address and information number:**

Press ADDR SRH or MSG NO. SRH, enter the address or information number to be searched, press enter to start searching, then locate the cursor to the searched address or information number.

(4) View and set initialized data table

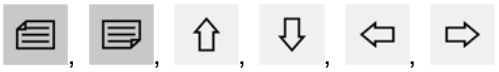
On edit page, press softkey INITDATA to enter to initialized data table display page:



	7	6	5	4	3	2	1	0
K0000	0	0	0	0	0	0	0	0
K0001	0	0	0	0	0	0	0	0
K0002	1	0	0	0	0	0	1	0
K0003	0	0	0	0	0	0	0	0
K0004	0	0	0	0	0	0	0	0
K0005	0	0	0	0	0	0	0	0
K0006	0	0	0	0	0	0	0	0
K0007	0	0	0	0	0	0	0	0
K0008	0	0	0	0	0	0	0	0
K0009	0	0	0	0	0	0	0	0
K0010	1	0	0	0	1	0	0	0
K0011	0	0	0	1	0	1	0	0

① **K parameter setting**

Press window1 K to select the window 1 whose corresponding K parameter is shown in the above figure.

Press  , to select a bit in the K parameter to be set or modified, the explanation for the bit in K parameter is displayed on the bottom area of the screen.

Press INPUT repeatedly, you can set the bit to 0 or 1.

② **Initialized data**

Press window2 InitData to enter the initData table display page corresponding to window 2.

③ **Data table modification and edit:**

Select the desired page by pressing  or  ; press  ,  ,  ,  to select the address or address value to be modified, the background of the selected value will turn to blue; press  and input values through numerical keys (press backspace key to delete), then press  key again to confirm the modification.

Note: The operation for modifying and editing the initialized data table is the same as the viewing and setting of symbol table. For details, please refer to the "3.4.3 (2) View and set symbol table".

3.4.5 PLC Program Transmission

PLC program transmission is subject to authority above the 2nd level.

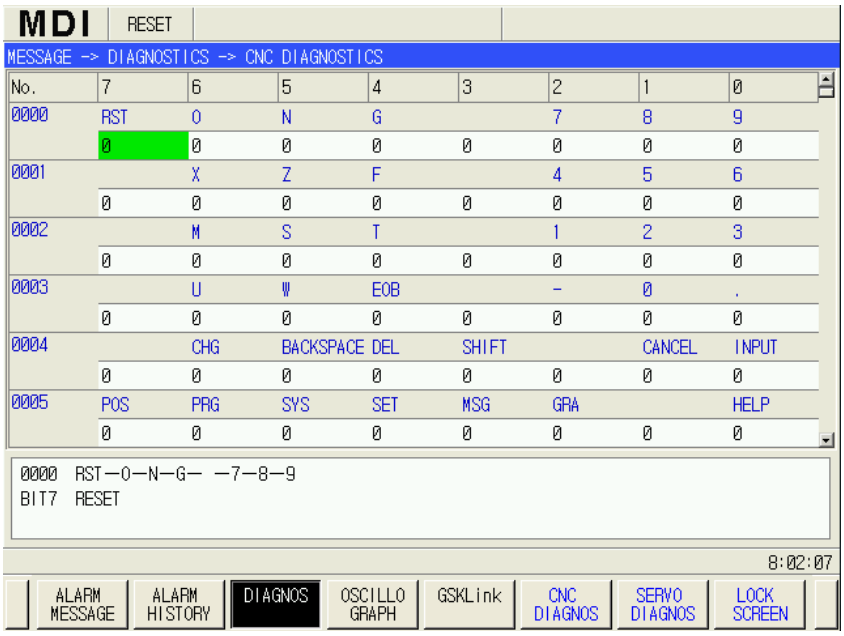
There are two method of PLC program transmission:

1. Transmit with GSKComm-M. For details, please refer to section 3.2 in this chapter.

2. Transmit PLC program one by one on PLC screen through U disk, or, make bulk transmission on file management screen. For details, please refer to section 3.3 in this chapter.

3.5 CNC Diagnosis

Press to enter to information screen, then press  to enter to diagnosis page, press  to enter to CNC diagnosis page.



On the system diagnosis page, contents such as keyboard diagnosis, state diagnosis, and auxiliary function parameter are included. They can be viewed through , , , , , . You can press LOCK SCREEN to lock to the current screen in case of mis-operation.

On CNC diagnosis display page, there are two lines at the bottom of the screen displaying the diagnosis details: the first line shows the diagnosis number; the second line shows the explanation of a bit in the parameter where the cursor locates.

The diagnosis information and corresponding number is shown as follows:

- **System keyboard diagnosis information (number: 0-7)**

It can diagnose all the keys on the system keyboard. Each key is in either pressed or released state. It is used to diagnose whether the keyboard is in good condition.

- **Feed axis diagnosis information (number:10-29)**

It includes the input/output state of servo drive unit connected with the feed axis, the pulse sent from feed axis to FPGA, the pulse sent from FPGA to servo drive unit, and the accumulative errors of the feed axis pulse (the difference between the FPGA received and sent pulse). It is used to diagnose whether the feed axis is in good working condition.

- **Pulse encoder diagnosis information (number:30~33)**

It includes the rotation direction of the two-channel pulse encoder, Z signal state, A,B phase signal state and the current counting pulse value. It is used to diagnose whether the pulse encoder is in good working condition.

- **MPG diagnosis information (number: 40-43)**

It includes the rotation direction of the two-channel MPG, A, B phase signal state and current counting pulse value. It is used to diagnose whether the encoder is in good working condition.

- **Spindle diagnosis information (number: 50~52)**

It includes the state of two-channel spindle alarm signal, tapping signal, enable signal and ready signal.

- **Machine tool panel diagnosis information (number: 60-62)**

It includes the accumulative errors number, consecutive errors number and repeated number of the machine tool panel. It is used to diagnose whether the machine tool panel is in good working

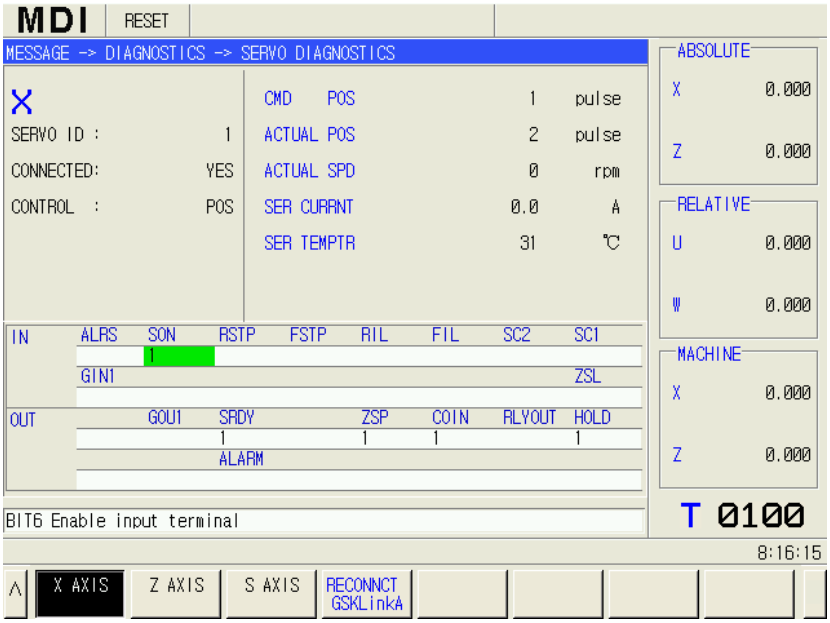
condition.

➢ **Edit keyboard diagnosis information (number: 63-65)**

It includes the accumulative errors number, consecutive errors number and repeated number of the edit keyboard. It is used to diagnose whether the edit keyboard is in good working condition.

3.6 Servo Diagnosis

On system screen, press DIAGNOS to enter to diagnosis page, then press SERVO DIAGNOS to enter to servo diagnosis page.



GSK988T servo diagnosis module provides the following functions:

It performs real-time monitoring to system controlled axes through servo communication feedback data, so the operator can know about the working state of some devices such as servo, motor etc.

(1) When the servo is in position control mode, the information displayed includes the command pulses received by the servo, the feedback pulses obtained from the motor encoder, actual rotation speed of the motor, servo internal current, detected temperature in the servo.

(2) When the servo is in speed control mode, the displayed information includes the specified rotation speed received by the servo, actual rotation speed obtained from the motor, command pulses received by the servo, servo internal current, detected temperature in the servo (spindle encoder value is displayed).

(3) The I/O sate when servo is connected with system.

Explanations for various data items on servo diagnosis screen:

X : Current selected axis name

SERVO ID : The number of the slave connected to the axis

CONNECTED : The connection state of servo communication link layer

CONTROL : The servo control mode

CMD POS : The position pulses received from the system (in position control mode)

CMD SPD: The speed command value received from the system (in speed control mode)

ACTUAL SPD: The position pulses feedbacked by the servo

ACTUAL POS: The actual rotation speed of the motor

ENCDER VAL: The current value of spindle encoder (in spindle or C axis control mode)

SER CURRNT: The servo working current value at present

SER TEMPTR : The detected temperature of the servo inside

IN : The servo input point value

OUT : The servo output point value

BIT6 Enable input terminal : Details of the servo input and output points where the cursor located

Axis switching: Press X AXIS, Z AXIS, S AXIS to switch the displayed servo parameter among X, Z, S axis

RECONNCT GSKLinkA : When some axes are not connected or the servo communication is erroneous, press it to reset the communication link. If the connection still cannot be done, turn on the power of servo and system again.

CHAPTER IV MACHINE DEBUGGING-FUNCTIONS

4.1 Emergency Stop and Hardware Limit

GSK988T is equipped with the software limit function; for safety, it is suggested to adopt the hardware limit function at the same time. Install the limit switches in positive and negative directions on axes; the connection is shown in the following figure:

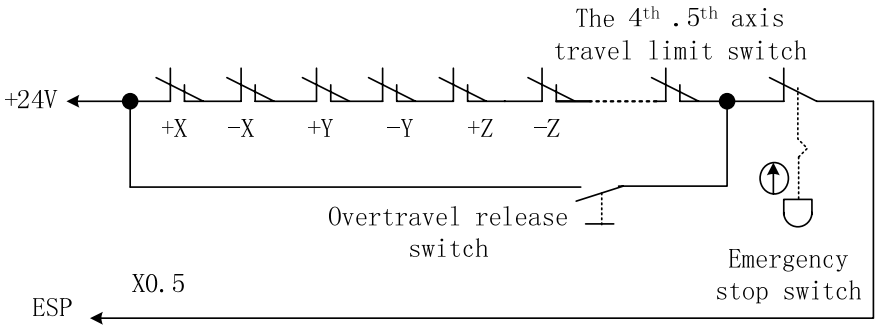


Fig. 4-1-1

In MANUAL or MPG mode, slowly move each axis to verify the validity of the overtravel limit switch, the correctness of the displayed alarm and the validity of the overtravel release button;

when it overruns or the emergency stop button is pressed, CNC alarms “emergency stop”. The alarm can be cleared through pressing the OVERTRAVEL and the axis moves in the reverse direction, or shield PLC emergency stop parameter which makes the switch invalid, and then press the resetting key to clear the emergency stop limit alarm, and the axis moves toward the worktable in the reverse direction and is off from the limit switch.

Relevant Parameter				
	Parameter No.	**Bit**	**Meaning**	**Remark**
CNC parameters	**3003**	**#7**	ESP emergency stop alarm signal (X0.5) 0: Emergency stop alarm occurs when the input signal is 0 (low-level signal) 1：Emergency stop alarm occurs when the input signal is 1 (high-level signal)	These two parameters should be set consistently.
Standard PLC parameters	**K0010**	**#7**	External emergency stop alarm signal (X0.5) 0: Emergency stop alarm occurs when the signal is low-level 1：Emergency stop alarm occurs when signal is high-level	

4.2 Basic Axis Parameters Setting

4.2.1 Axis Property

(1) Axis name

The controlled axis number of 988T is 2, the extended controlled axis number is 5 (including Cs axis), and the linkage controlled axis number is 3.

Relevant Parameter		
No.	**Meaning**	**Remark**
8130	Total controlled axis	
1010	CNC controlled axis number	This parameter value cannot be greater than parameter No.8130
1020	Name of each axis	The axis names cannot be the same
1022	Axis properties in basic coordinates	
1023	Servo axis number of each axis	The set axis number should correspond to the axis number of the rear cover interface.

(2) Axis unit

After the above parameters are set, you can set the axis names, input/output unit (metric or inch) and incremental system as requirement.

Relevant Parameters		
No.	**Bit**	**Meaning**
0000	**#2**	Input unit 0: metric 1: inch
1001	**#0**	The least movement unit of the linear axis: 0: metric (metric machine) 1: inch (inch machine)
1004	**#1**	Set the least input increment and least command increment 0: 0.001mm 1: 0.0001mm
1006	**#3**	Set the movement amount of each axis 0: by radius 1: by diameter
3401	**#0**	For the addresses which decimal points can be used, when the decimal point is ignored, the setting is as follows: 0: as the minimum setting unit 1: as mm, inch, s

(3) Axis type

During machining, you can set the axis as linear axis or rotary axis according to the machining requirement.

Relevant Parameter		
No.	**Bit**	**Meaning**
1004	**#6**	Whether the least input increment of rotary axis is 10 times of the least command increment: 0: No 1: Yes
1006	**#1, #0**	00: Linear axis 01: rotary axis (A Type) 11: rotary axis (B type)

1008	**#0**	Whether the circle display function of rotary axis is valid: 0: invalid 1: valid
1008	**#1**	The rotation direction of absolute command is : 0: the direction closer to the object 1: specified by command value symbol
1008	**#2**	Relative coordinate is : 0: does not cycle by the movement amount per rotation 1: cycles by the movement amount per rotation
1260		Movement amount per rotation of each rotary axis

4.2.2 GSKLink Communication Setting of Axis and Servo

When GSKLink communication between feed and spindle servo drive unit is to be executed, the servo slave number and communication baudrate need to be set.

Note 1: The slave number corresponding to the servo communication cannot be repeated. The baudrate of servo should be consistent with the system communication baudrate; otherwise, the communication cannot be set up.

Note 2: The parameters and slave number and baudrate in the servo should be set before the servo is connected to the GSKLink, and should be modified manually in the drive unit (valid after power-on again).

Relevant Parameter					
	Type	**No.**	**Bit**	**Meaning**	**Remark**
CNC	**GSK 988T System parameter**	**9000**	**#0**	System servo communication function is : 0: invalid 1: valid	
		9010		System servo communication baudrate	Be consistent with the servo baudrate
		9011		Slave number corresponding to the servo communication of each axis	The slave number cannot be repeated.
		9012		Slave number of analog spindle communication (SIDS1)	
Servo	**DAT 2000C Series**	**PA58**		GSKLink communication servo slave number	Corresponds to the CNC parameter No. 9011
		PA59		GSKLink communication baudrate	Be consistent with the CNC parameter No. 9010
		PA61		Drive unit type	101: 2050C; 102: 2075C
	DAP03C DAY3025C	**PA19**		GSKLink communication servo slave number	Corresponds to the slave number set by CNC parameter No. 9012, 9013
		PA20		GSKLink communication baudrate	Be consistent with the baudrate set by CNC parameter 9010

4.3 Servo Related Setting

4.3.1 CNC Servo Parameter Setting

After the connection between system and servo is done, and the power is on, you can set the high-level or low-level servo alarm, encoder type, pulse output type, pulse output direction of each axis and axis movement direction. The procedure is as follows:

① According to the alarm logic level of servo drive unit set corresponding alarm level of servo axis through parameter NO.1816#0 (ISAx).

② Select the pulse output method of current axis through parameter No.1811#0 (ABPx). GSK988T supports two methods of pulse command output, one is + direction pulse output; the other is orthogonal two phases (AB phase) pulse output. The parameter should be set correctly.

③ According to the encoder type of the servo motor set whether absolute encoder is used on servo through parameter No. 1815#5 (APCx).

④ According to the transmission ratio of the machine set correct gear ratio CMR/DMR (through parameter No. 1816 and No. 1820) to make the movement distance be consistent with the specified value (for details, see follow-up sections).

⑤ When the machine movement direction is inconsistent with the movement command, inverse the output movement direction of servo axis command by setting parameter No. 1811#2 (PODx).

⑥ When the detection direction of absolute encoder is opposite to the actual direction, adjust it through parameter No.1815#0 (APRx). (This detection method is: when the reference point is set, specified axis moves along + direction +U, and +U is displayed in the machine coordinate system, turn on the power again; if –U is displayed in the machine coordinate system, then, the detection direction of absolute encoder is opposite to the actual direction.)

Note 1: In AUTO or MANUAL mode, when the specified axis direction is opposite to the actual feeding direction, modify parameter No. 1811#0.
Note 2: In manual mode, when the manual feed direction is opposite to the actual feeding direction (correct in AUTO or EDIT mode), modify PLC data parameter K8.0~K8.4.

Relevant Parameter			
	No.	**Bit**	**Meaning**
GSK 988T System parameter	**1811**	**#0**	Pulse output type: 0 : pulse + direction 1: AB phase pulse
	1811	**#2**	Pulse output direction: 0: Not reversed 1: reversed
	1815	**#0**	When absolute encoder is used, the direction of position detector: 0: Not reversed 1: reversed
	1815	**#5**	Position detector is: 0: absolute position detector 1: not absolute position detector
	1816	**#0**	Servo alarm signal level: 0: high-level alarm 1: low-level alarm
PLC data parameter	**K8.0～K8.4**		Manual movement direction of each axis is: 0: not reversed 1: reversed

4.4 Gear Ratio Adjustment

Electronic gear ratio enables the move distances of machine tool slide be consistent with the specified distance (move distance of the machine coordinate). If the axis is set in radius system (radius or diameter programming is set by parameter No.1006#3), the actual move distance of the machine equals to the move distance displayed on the machine coordinate system; if the axis is set in diameter system, the actual move distance of the machine is the double of the move distance displayed on the machine coordinate system.

Please note that the electronic gear ratio is related to both the setting of CNC gear ratio and servo drive unit gear ratio.

Relevant Parameters		
No.	**Bit**	**Meaning**
0000	**#2**	Input unit: 0: metric 1: inch
1004	**#1**	Least input increment and least command increment: 0: ISB system 1: ISC system
1006	**#1、#0**	Linear axis or rotary axis 00: linear axis 01: rotary axis (A type) 11 : rotary axis (B type)
1006	**#3**	Movement amount of each axis is 0: specified by radius 1: specified by diameter
1816	**#6、#5、#4**	Set the detect multiplying ratio of each axis
1820		Set the command multiplying ratio of each axis

Incremental System						
			IS－B		IS－C	
	Input	**Radius/ Diameter**	**Least setting increment**	**Least command increment**	**Least setting increment**	**Least command increment**
Metric machine	**Metric**	**Diameter**	0.001mm	0.0005mm	0.0001mm	0.00005mm
		Radius	0.001mm	0.001mm	0.0001mm	0.0001mm
	Inch	**Diameter**	0.0001 inch	0.0005mm	0.00001 inch	0.00005mm
		Radius	0.0001 inch	0.001mm	0.00001 inch	0.0001mm
Inch machine	**Metric**	**Diameter**	0.001mm	0.00005 inch	0.0001mm	0.000005 inch
		Radius	0.001mm	0.0001 inch	0.0001mm	0.00001 inch
	Inch	**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
Rotary axis			0.001deg	0.001deg	0.0001deg	0.0001deg

4.4.1 Gear Ratio Calculation

Formula

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

Note: Least command increment is the minimum unit of command from CNC to machine tool, and the minimum increment of tool movement.

Pulse/rev = Encoder line number (absolute encoder)

= 4 × Encoder line number (incremental encoder)

Z_M ： Teeth number of lead screw gear

Z_D ： Teeth number of motor gear

Example：

When a machine is equipped with GSK988T and DAT2050C, and the ISC system is applied, the X axis is programmed in diameter system, the lead is 6mm, Z axis is programmed in radius system, the lead is 8mm; the motor is connected with X, Z axis lead screw directly (Z_M： Z_D=1： 1); 17-bit absolute encoder is applied (the encoder line number is 2^{17} , i.e. 131072), the calculation of corresponding gear ratio of X, Z axis is as follows:

X axis:

Least command increment: 0.00005mm (ISC system, programmed in diameter)

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

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

Z axis:

Least command increment: 0.0001mm (ISC system, programmed in radius)

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

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

4.4.2 Gear Ratio Setting

Parameter Setting of Gear Ratio

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

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

When the numerator is greater than the dominator in CNC electronic gear ratio (CMR/DMR), the CNC permitted maximum speed will be decreased; when the numerator is smaller than the dominator, the CNC position accuracy will be decreased. To ensure the target accuracy and speed, when digital servo with electronic gear ratio function is matched, it is advised to set the CNC electronic gear ratio

to 1:1, and set the calculated electronic gear ratio into digital servo.

Example: (the gear ratio is the one in the example of Gear Ratio Calculation)

X axis

CNC gear ratio setting

CNC electronic gear ratio is set as 1:1, i.e. CMR/DMR=1:1

Setting value of CMR (Parameter No. 1820) is 2.

Setting value of DMR (Parameter No.1816)(**DM3x**：**DM2x**：**DM1x**) is 001.

Servo gear ratio setting

The servo gear ratio is set as 2048 / 1875.

Setting value of PA12 is 2048.

Setting value of PA13 is 1875.

Z axis

CNC gear ratio setting

CNC electronic gear ratio is set as 1:1, i.e. CMR/DMR=1:1

Setting value of CMR (Parameter No. 1820) is 2.

Setting value of DMR (Parameter No.1816)(**DM3x**：**DM2x**：**DM1x**) is 001.

Servo gear ratio setting

The servo gear ratio is set as 1024 / 625.

Setting value of PA12 is 1024.

Setting value of PA13 is 625.

4.5 Acceleration/Deceleration Characteristic Adjustment

As the acceleration/deceleration time constant increases, the acceleration/deceleration process slows down, the impact to machine tool decreases, and the machining efficiency decreases; and vise versa.

When the time constant is the same, the higher the start/end speed of acceleration/deceleration is, the greater the impact will be, so are the machining efficiency; and visa versa.

The principle of acceleration/deceleration characteristics adjustment is: to properly reduce the acceleration/deceleration time constant and increase the start/end speed of acceleration/deceleration to improve processing efficiency in the condition that the drive unit won't issue an alarm, the motor can work without the loss of steps and no obvious impact occurs to the machine tool. If the acceleration/deceleration time constant is too small or the start/end speed is set too high, it can easily lead to drive unit alarm, motor step loss or machine tool vibration.

Note: when parameter No.1601#4=0, at the intersection point of cutting feed paths, the feedrate should be decreased to the start speed of the acceleration/deceleration, then, increased to the specified speed of the adjacent block. Accurate position of intersection point can be achieved in this way, but it may lower down the machining efficiency.

When parameter No.1601#4=1, two adjacent cutting paths perform smooth transition in acceleration/deceleration method directly. The feedrate does not necessarily decrease to the start speed when the previous path ends. An arc transition is formed at the intersection point (inaccurate position). This kind of transition way allows great surface smoothness and higher machining efficiency.

Relevant Parameter		
No.	**Bit**	**Meaning**
1420		Rapid traverse rate of each axis
1421		The minimum speed of rapid traverse override
1422		Maximum cutting feedrate of all axes
1423		Manual feedrate of each axis
1424		Manual rapid traverse rate of each axis
1601	**#4**	During rapid traverse, the blocks are: 0: not overlapped (accurate) 1: overlapped (smooth transition)
1610	**#0**	The acceleration/deceleration of cutting feedrate (including dry run feeding) is: 0: exponential type 1: linear type after interpolation
1610	**#4**	The acceleration/deceleration of manual feeding is : 0: exponential type 1: linear type or bell-shaped type after interpolation
1620		Constant T during linear acceleration/deceleration of each axis
1622		Acceleration/deceleration time constant of cutting feedrate after interpolation
1623		Exponential acceleration/deceleration FL speed of cutting feedrate
1624		Acceleration/deceleration time constant of manual feedrate after interpolation
1625		Exponential acceleration/deceleration FL speed of manual feedrate
1626		Acceleration/deceleration time constant during thread cutting cycle
1627		Exponential acceleration/deceleration FL speed during thread cutting cycle

4.6 Reference Point and Software Limit

GSK988T supports three kinds of method to set machine zero point (also called reference point): reference point setting without dogs, reference point setting with dogs and absolute encoder reference point setting.

Reference Point Setting	System Parameter Setting
absolute encoder reference point setting	Parameter No.1815#5 (APCx) is set to 1
reference point setting without dogs	Parameter No.1815#5 (APCx)is set to 0 Parameter No.1002#1 (DLZ) is set to 1 or parameter No.1005#1 (DLZx) is set to 0 (either one of them is set to 1)
reference point setting with dogs	Parameter No.1815#5 (APCx) is set to 0 Parameter No.1002#1 DLZ is set to 0 and parameter No.1005#1 DLZx is set to 0.

Note 1: When absolute encoder is used, after a reference point is set, it will be saved automatically after power-off, so it is not necessary to set the reference point the next time.

Note 2: When reference point is set with/without dog, the setting should be executed every time after power-on.

Relevant Parameter			
	No.	**Bit**	**Meaning**
GSK988T System Parameter	**1005**	**#0**	When the reference point is not set, in AUTO (MEM, DNC OR MDI) mode, if movement commands other than G28 is specified, the system will 0: issue an alarm 1: not issue an alarm
	1006	**#5**	Set reference point return direction of each axis 0: positive direction 1: negative direction
	1201	**#2**	After the manual reference point return is done, the local coordinate system is: 0: not cancelled 1: cancelled
	1240		Set the coordinate value of the 1st reference point in machine coordinate system
	1241		Set the coordinate value of the 2ndt reference point in machine coordinate system
	1242		Set the coordinate value of the 3rd reference point in machine coordinate system
	1243		Set the coordinate value of the 4th reference point in machine coordinate system
	1425		Set the FL speed after deceleration during reference point return
PLC Data Parameter	**K12.2**		Whether the direction keys of zero-return operation is automatically locked: 0: No 1: Yes

4.6.1 Reference Point of Absolute Encoder Setting

When the machine is equipped with the absolute position encoder and its reference point return function is valid, it requires setting the absolute position encoder reference point while the system doesn't set the reference point or readjust the reference point. After setting the reference point, the system automatically saves the reference point position after power-off, therefore, it doesn't require setting the reference point position, again when it powers on in the next time.

The steps of setting the reference point of absolute encoder are shows as follows:

1. In mode of 

or 

mode, the axis carriage is moved to the reference point to be set.

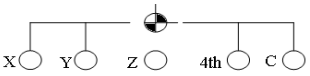
2. The system parameter APZx (NO.1815#4)is set as 0, cut off power supply, and power on, again, the system alarms.

3. Press 

on the panel.

4. Press the switches 

related to the reference points, then the reference point return is performed.

5.,The reference point return finishing indicator (operational panel indicator) 

is on, the reference point return finishes, the system saves the current encoder position, automatically sets the parameter APZ x as 1.

Relevant Parameter		
No.	**Bit**	**Meaning**
1815	**#0**	When absolute encoder is used, the direction of position detector is: 0: not reversed 1: reversed
1815	**#4**	When absolute encoder is used, the machine position and the position of absolute position detector is 0: inconsistent 1: consistent
1815	**#5**	Position detector: 0: uses absolute position detector 1: does not use absolute position detector

4.6.2 Reference Point Setting with Dog

When the reference point return without dogs function is set invalid (i.e. the reference point return with dogs is valid), reference point return can be performed when the machine is installed deceleration switch. After the tool returns to the reference point, LED indicator lights up, and coordinate system is set automatically.

The procedure of setting the reference point free of block is shown as follows:

1. Confirm that the overtravel limit switch is valid.

2. Press 

to switch the working mode to reference point return mode.

3. To reduce zero-return speed, adjust the rapid traverse override 

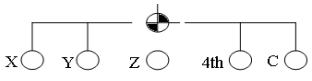
to the low gear.

4. Press the corresponding axis icon on 

, the system will read axis and the direction selection signal, then, reference point return is executed.

5. When the carriage moves to the deceleration point at the rapid traverse rate, and the deceleration signal DECx is valid (the valid level of a signal is set by parameter No. 3009#5 DECx), the speed will decrease to 0, then, the carriage keeps moving toward the reference point at the speed set by parameter No1425.

6. When the tool leaves the position of deceleration switch, set the deceleration signal DECx to 1, and the system detects the one-rotation signal nPC.

7. After the first nPC signal is detected, set the signals ZPx (reference point return finish signal) and ZRFx (reference point setting signal) to 1, the reference point return indicator 

lights up, reference point return is done.

Note: Usually, the machine zero return dog is installed at the maximum stroke point. The effective stroke should be over 25mm to ensure enough deceleration distance and the accuracy of zero return. The higher the machine zero return speed is, the longer the zero return collision block will be. Otherwise, the carriage will go through the collision block due to CNC acceleration/deceleration or machine inertial, thus, affects the accuracy of zero return. In addition, make sure that during the process of zero return the carriage will not be intervened by other parts of the machine tool, for the sake of security.

The connection method of AC servo motor is shown in the following figure. Stroke switch and servo motor one-rotation signal are used.

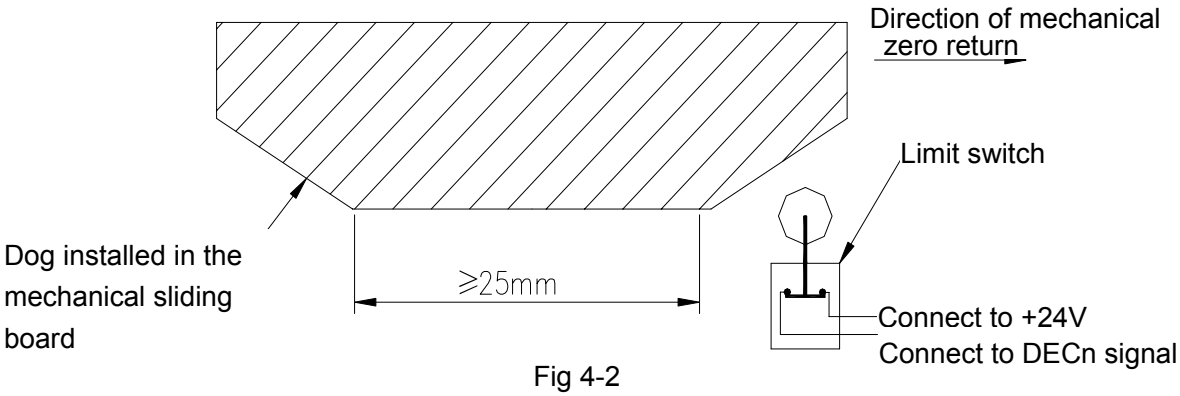


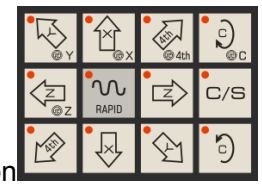
Fig 4-2

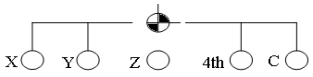
When machine zero return is performed after the deceleration switch is released, it should be noted that the encoder one-rotation signal should not be at the critical point, and be reached after half-revolution of the motor. This method is to improve the machine zero point return accuracy. The dog position can be tuned to reduce the error of zero point return.

4.6.3 Reference Point Setting without Dog

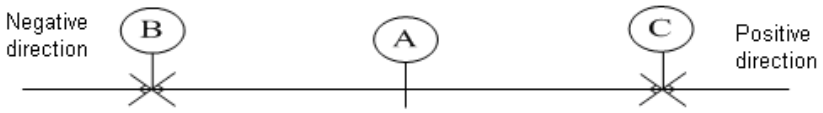
When the system sets reference point without dog function valid, the machine can perform reference point return without the installation of deceleration switch. After reference point return is done, the LED indicator lights up and the coordinate system is set automatically.

1. Axis moves along the reference point return direction, and stops close to the reference point, rather than surpasses it.
2. Press 

and set the reference point selection signal ZRN to 1 manually;
3. Select corresponding feeding axis and direction on 

, and set the axis and direction signal Jx to 1, reference point return is executed.
4. The tools moves towards the reference point along the direction set by parameter No. 1006#5 at the speed set by parameter No. 1425.
5. After the first PC signal is detected, set ZPx (reference point return end signal) and ZRFx (reference point setting signal) to 1, the LED indicator 

lights up and reference point return is done.



A: the position before the execution of reference point return without dog

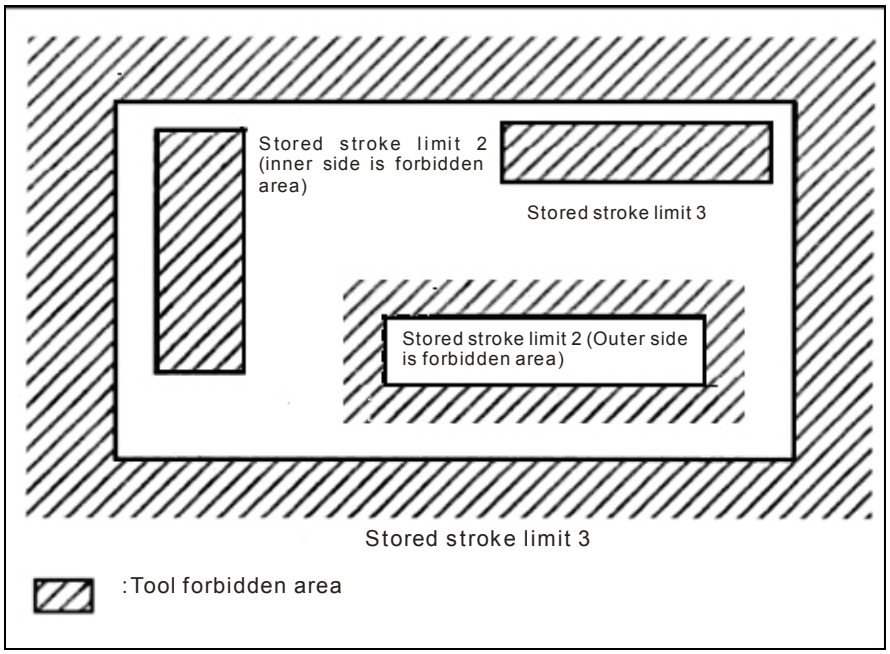
B: the position after the reference point return is executed along the negative direction, i.e. the position where the first PC signal is generated after the A point moves along the negative direction.

C: the position after the reference point return is executed along the negative direction, i.e. the position where the first PC signal is generated after the A point moves along the negative direction.

Relevant Parameters		
No.	**Bit**	**Meaning**
1002	**#1**	The function of reference point return without dog is 0: invalid 1: valid (for all axes)
1002	**#3**	The G28 command when reference point is not set: 0: reference point return with dog 1: P/S alarm occurs
1005	**#1**	The function of reference point setting without dog is: 0: invalid 1: valid
1300	**#6**	Whether the first stroke check is performed before manual reference point return after LZR power-on: 0: Yes 1: No

4.6.4 Setting of Stored Stroke Check

There are three stored stroke check are provided in GSK988T system: stored stroke check 1, stored stroke 2 and stored stroke 3. Tool cannot enter the areas specified by them.



Stored stroke check 1:

Parameters (No.1320, 1321 or No. 1326, 1327) set the boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area to maximum stroke.

Stored stroke check 2: (G22 G23)

Parameters (No.1322, 1323) or commands set the boundaries. Inside or outside the area of the limit can be set as the forbidden area. Parameter No. 1300#0 selects either inside or outside as the forbidden area. In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area. Each of G22 and G23 should be command independently of another command in a block.

Stored stroke check 3:

Parameters No. 1324, No.1325 set the boundary. The inside area of the 3 limits is forbidden area.

Note：If the two points for specifying a forbidden area are identical, all areas are handled as forbidden areas for stroke check 1.
If two points for specifying a forbidden area are identical, all areas are handled as movable areas for stored stroke check 2, 3.

Alarm displaying time:

Parameter No.1300#7 (BFA) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediate after the tool has entered the forbidden area.

Releasing the alarm:

If the tool enters a forbidden area and an alarm is generated, the tool can be moved only in the backward direction. To cancel the alarm, move the tool backward until it is outside the forbidden area and reset the system. When the alarm is cancelled, the tool can be moved both backward and forward.

Relevant Parameter		
No.	**Bit**	**Meaning**
1300	**#0**	Parameters (No.1322, 1323) set the forbidden area for stored stroke check 2 to: 0: inside area 1: outside area
1300	**#2**	The signal EXLM (stored stroke check switch signal) is : 0: invalid 1: valid
1300	**#5**	Stroke check 3 releasing signal RLSOT 3 is: 0: invalid 1: valid
1300	**#6**	Whether the first stroke check is performed before manual reference point return after power-on: 0: Yes 1: No
1300	**#7**	The alarm is issued: 0: before the tool enters the forbidden area 1: after the tool enters the forbidden area
1310	**#0**	Whether stored stroke check 2 is performed on each axis: 0: no 1: yes
1310	**#1**	Whether stored stroke check 3 is performed on each axis: 0: no 1: yes
1320		Coordinates (PC1) of stored stroke check 1 positive boundary
1321		Coordinates (NC1) of stored stroke check 1 negative boundary
1322		Coordinates (PC2) of stored stroke check 2 positive boundary
1323		Coordinates (NC2) of stored stroke check 2 negative boundary
1324		Coordinates (PC3) of stored stroke check 3 positive boundary
1325		Coordinates (NC3) of stored stroke check 3 negative boundary
1326		Coordinates II (PC12) of stored stroke check 1 positive boundary
1327		Coordinates II (NC12) of stored stroke check 1 negative boundary

4.7 Pitch Error Compensation

If the pitch error compensation value is defined, the pitch error compensation of each axis can be compensated based on the detection units of each axis.

Set the pitch error compensation data for each compensation position, and its compensation position is set based on the space between each axis. The compensation origin is the reference position of the tool return.

When the pitch error compensation is performed, the following parameters must be set:

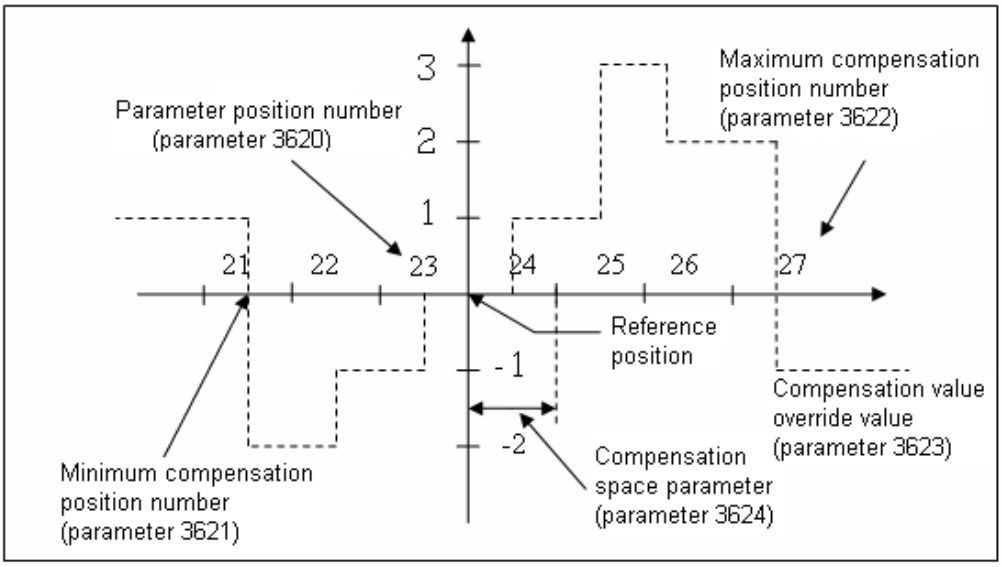
Parameter 3620: The position number of the compensation at the reference point of each axis.

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

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

Parameter 3623: The magnification of pitch error compensation

Parameter 3624: The interval of each compensation position.



Compensation Position Number	21	22	23	24	25	26	27
Set Compensation Value	-3	+1	+1	+1	+2	-1	-3

Define the compensation position: To set the compensation position for each axis, specify the positive or negative direction for compensation based on the reference point. If the machine stroke exceeds the specified range in positive or negative direction, the screw pitch error compensation does not work.

Compensation position number: In screw pitch error compensation setting screen, there are 1024 compensation positions (0~1023) can be used. The parameter can be used to assign position number to each axis. Set compensation position number (parameter No.3620), minimum position number (parameter No. 3621) and maximum position number (parameter No. 3622) of each axis.

For example:

1. Linear axis

Machine stroke：－400mm~＋800mm

Interval of the screw pitch error compensation positions：50mm

Compensation position number of the reference point: 70

After the above definition is finished, the furthest compensation position number in negative direction is as below:

The compensation position number of the reference point – (machine stroke in negative direction/space between compensation positions) = 70－400/50+1=63

The furthest compensation position number in positive direction is as below:

Compensation position number of the reference point + (machine stroke in positive direction/ space between compensation positions) ＝70＋800/50=86

The corresponding relation between the machine and the compensation point position number is shown as follows:

Parameter	Setting Value
3620: Parameter point compensation number	70
3621: Minimum compensation position number	63
3622: Maximum compensation position number	86
3623: Compensation magnification	1
3624: Space between the screw pitch error compensation positions	50000

2. Rotary axis

Movement value/ revolution: 360°

Space between the screw pitch error compensation positions: 45°

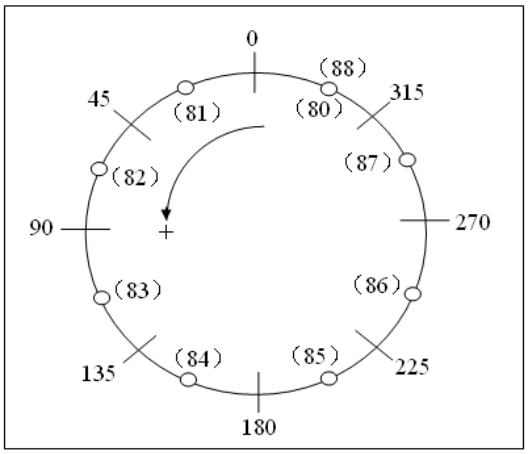
Compensation position number of the reference point: 80

After defining the above parameter, the furthest compensation position number in negative direction of the rotary axis is the compensation position number of the reference point.

The furthest compensation position number in positive direction is as follows:

The compensation position number of the reference point + (movement value of each revolution/ space between compensation positions) = 80+360/45=88

The corresponding relation between the machine coordinate and the compensation position number is as follows:



The parameter is set as follows:

Parameter	**Setting value**
3620: Compensation number of the reference point	80
3621: Minimum compensation position number	80
3622: Maximum compensation position number	88
3623: Compensation magnification	1
3624: Space between screw pitch error compensation positions	45000

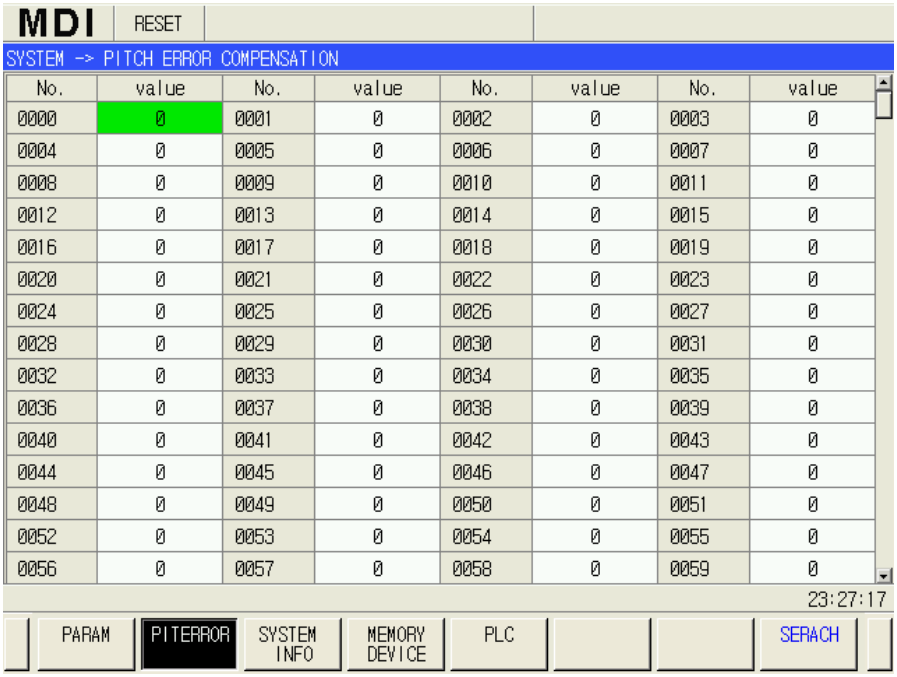
For the rotary axis, there may result in the position offset if the sum of the compensation value of positions 81~88 is not 0. The sum is the accumulation of screw pitch error compensation value of each revolution. Moreover, at the compensation positions 80 and 88, the same compensation value must be set.

For example:

No. of Compensation Position	80	81	82	83	84	85	86	87	88
Set Compensation value	+1	-2	+1	+3	-1	-1	-3	+2	+1

Set pitch error compensation value

- In system window, press PITERROR to enter into pitch error compensation window, shown as follows:



You can view and set corresponding pitch error compensation value in this window.

- Press , , or move the cursor through , , , , to select the value to be set; or, press SERACH to search for the pitch error compensation number, and move the cursor to the value to be modified.
- Press INPUT, the selected compensation can be modified, input the desired value, then press INPUT to complete the modification.

Note: The compensation value and interval of compensation point are related to the programming method (diameter programming/radius programming affects the least command increment). When the axis movement is programmed in diameter, the parameter value should be set in diameter; when the axis movement is programmed in radius, the parameter value should be set in radius. The unit should be detection unit.

Relevant Parameter		
No.	**Bit**	**Meaning**
3620		Pitch error compensation number of reference points
3621		Pitch error compensation number of the farthest ends in negative direction
3622		Pitch error compensation number of the farthest ends in positive direction
3623		Pitch error compensation magnification
3624		Intervals of compensation points
3628	**#4~#0**	Setting value of pitch error compensation pulse frequency (in the form of BCD code)

4.8 Backlash Compensation

When the machine tool moves backward inverse momentum loss will occur due to the error of transmission mechanism, thus affects the machining accuracy. To reduce such error, backlash compensation function is provided in this system.

The backlash compensation value is related to the programming method (diameter programming/radius programming affects the least command increment). When the axis movement is programmed in diameter, the parameter value should be set in diameter; when the axis movement is programmed in radius, the parameter value should be set in radius. The unit should be detection unit.

Detection Unit = Least command increment/command multiplication (CMR)

The backlash compensation should be performed in a proper way to improve the machining accuracy. It is advised to use dialgauge, micrometer or laser detector rather than MPG or step method to measure the backlash. The methods are shown as follows:

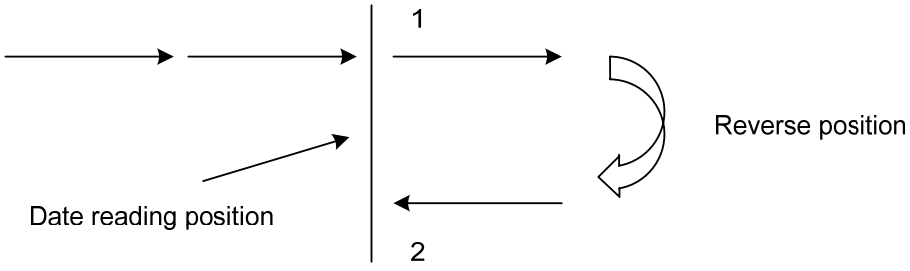
The setting method of backlash compensation during cutting feed

Programming:

O0001;
N10 G01 W10 F800;
N20 W15;
N30 W1;
N40 W-1;
N50 M30.

Set the backlash compensation value to 0 before measuring

Run a single block and find the measuring benchmark 1 after twice locations, record the current data, then, move further for 1mm and move backward for 1mm to benchmark 2, read the current data.



Backlash compensation value=| data at benchmark 1- data at benchmark 2| ; then, convert the

calculated data to detection unit and input the CNC data parameter No. 1851.

Data 1: the dialgauge data read at benchmark 1

Data 2: the dialgauge data read at benchmark 2

Detection unit= Least command increment/CMR

For example:

When IS-B system is set (parameter No.1004#1 ISC is set to 1) and the metric system is selected (parameter No.1001#0 INM is set to 0), if parameter No. 1820 (used to set the command multiplication) is set to 2, then, the system command multiplication CMR=1.

X axis: detection unit=least command increment/CMR=0.00005mm/1=0.00005mm

Z axis: detection unit=least command increment/CMR=0.0001mm/1=0.0001mm

If the backlash compensation value of X axis detected by dialgauge is 0.0150mm, the parameter No. 0851 is set to 300; If the backlash compensation value of Z axis detected by dialgauge is 0.0300mm, the parameter No. 0851 is set to 300;

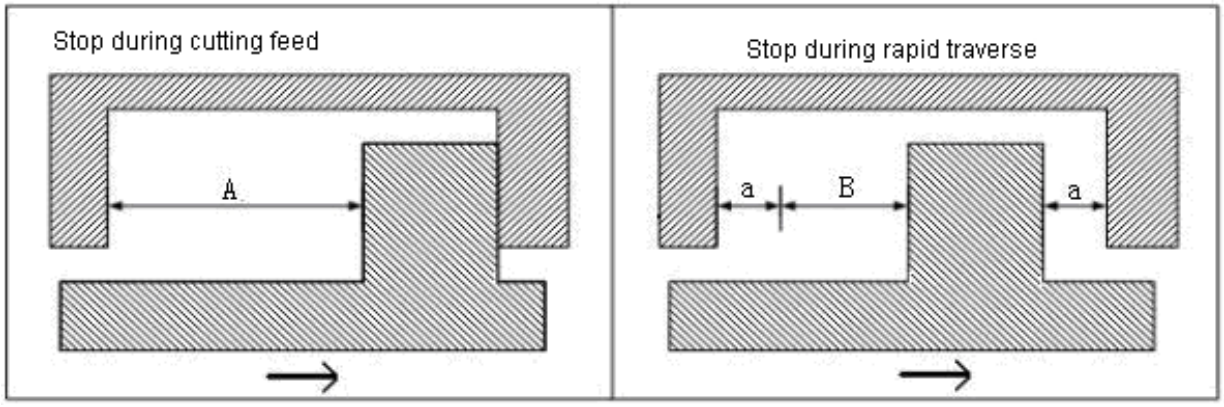
To improve the compensation accuracy, the backlash compensation value can also be set to rapid traverse and cutting feed. First, set parameter No.1800#4 (RBK) to 1, (cutting feed and rapid traverse will be performed independently), then, set the backlash compensation value to rapid traverse through parameter No.1852.

For example,

If the detected backlash compensation during cutting feed is A, and the detected backlash compensation during rapid traverse is B, according to different feeding method and move direction, the compensation value is shown in the following table:

Feeding method / Move direction	Cutting feed to cutting feed	Rapid traverse to rapid traverse	Rapid traverse to cutting feed	Cutting feed to rapid traverse
The same direction	0	0	±a	± (-a)
Opposite direction	±A	±B	± (B+a)	± (B+a)

1. a= (A-B)/2
2. The positive or negative of the compensation value determines the move direction. (P80)



The setting steps of backlash compensation parameters are shown as follows:

① Whether the backlash compensation is performed respectively during cutting feed and rapid traverse determines the setting of parameter No.1800#4 (RBK).

0: not performed 1: performed

② Measure the backlash compensation value in the above method, and save the results in parameter No. 1851 and No. 1852 (when parameter No. 1800#4 RBK is set to 1). Note that the parameter unit should be detection unit.

③ After the backlash compensation value is set, set the backlash compensation output method according to parameter No. 1800#7 (BDEC), 0: fixed pulse frequency output 1: output according to acceleration/deceleration characteristics. No. 1800#4.

④ When the parameter No. 1800#7 (BDEC) is set to 0 (fixed pulse frequency output), parameter No. 1800#6 (BD8) sets the output pulse frequency.
0: the set frequency 1: 1/8 of the set frequency.
The set frequency for compensation is set by parameter No. 1853.

⑤ When parameter No. 1800#7 (BDEC) is set to 1 (according to acceleration/deceleration characteristic output), the valid time constant can be set by parameter No. 2071.

Relevant Parameter		
No.	**Bit**	**Meaning**
1800	**#4**	Whether backlash compensation is performed respectively during cutting feed and rapid traverse 0: No 1: Yes
1800	**#6**	Output pulse frequency for backlash compensation is 0: the set frequency by parameter No. 1853 1: 1/8 of the set frequency.
1800	**#7**	Backlash compensation method 0: fixed pulse frequency output (set by parameter No. 1853 and No.1800#6) 1: output according to acceleration/deceleration characteristic
1851		Backlash compensation value
1852		Backlash compensation value during rapid traverse
1853	**#0~#4**	Setting value of backlash compensation pulse frequency
2071		Valid time constant of backlash compensation acceleration/deceleration

4.9 Spindle Function Adjustment

4.9.1 Spindle Encoder

GSK988T has two-channel encoder input interfaces (CN21 and CN22). CN21 interface is used for feedback input of spindle speed by default. The selection signal PC2SLC (G28.7) of spindle encoder in PLC selects the interface through which the feedback pulse is obtained and used for system control. When encoder interface 2 (CN22) is not connected to a encoder and the selection signal PC2SLC of position encoder is not set to 1, CN21 is always selected for the feedback input of spindle speed.

To read the actual spindle speed, relevant parameters and signals should be set correctly.

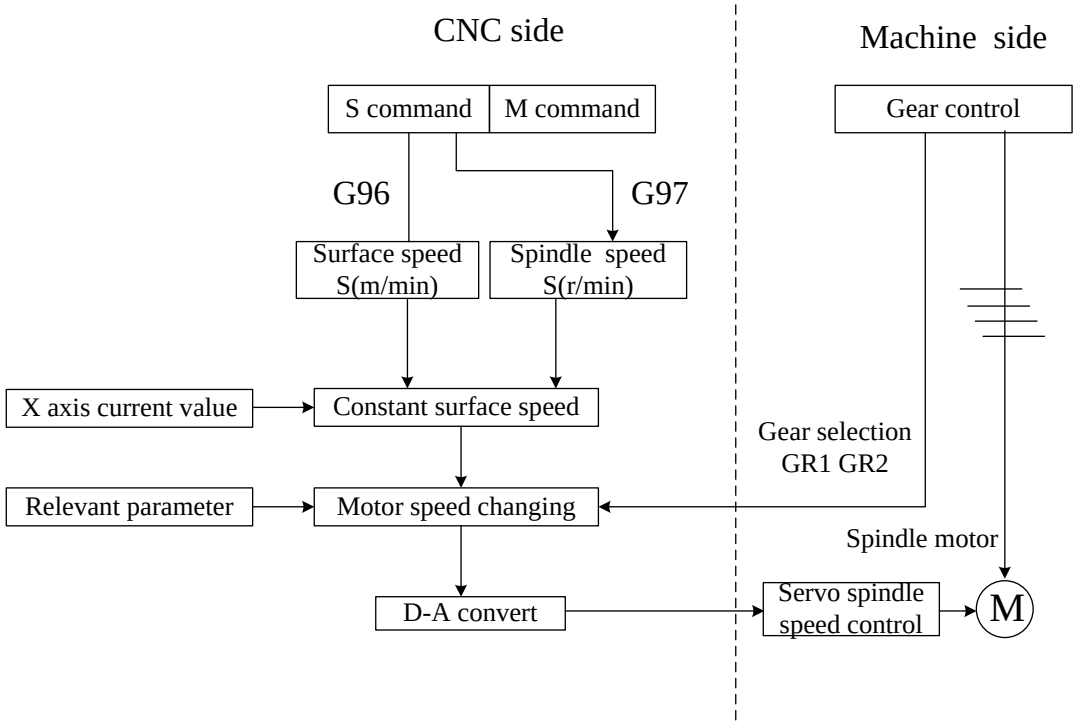
Relevant Parameter		
No.	**Bit**	**Meaning**
3706	**#1、#0**	Gear ratio between spindle encoder and position encoder 00：1　01：2　10：4　11：8
3707	**#1、#0**	Gear ratio between spindle encoder and the 2nd position encoder 00：1　01：2　10：4　11：8
3773		Spindle encoder line number　(CNT)
3803		The 2nd spindle encoder line number　(CNT2)

4.9.2 Spindle Speed Analog Voltage Control

Spindle speed analog voltage control can be set through CNC parameters. The interface outputs 0~10V analog voltage to control spindle servo drive unit or Inverter. For the 0V~+10V output control, the CNC calculates the spindle speed through S command and send M code to PLC to determine the spindle output direction.

Although the S command is for spindle speed, its actual control object is spindle motor. Therefore, the spindle motor speed and gear are related. In this system, the gear selection signal (GR1, GR2) determines the current gear on the machine; CNC outputs the spindle speed corresponding to the gear.

The spindle speed control procedure is shown as follows:



When the actual spindle speed is inconsistent with the programmed one, adjust it through parameter No. 3730, No. 3731. There are two adjust methods according to spindle encoder connection.

1．The spindle encoder is not used:

① Set the parameter No. 3730 to 1000 (gain adjustment data of spindle speed analog output) and parameter No. 3731 to 0 (compensation value of spindle speed analog output offset voltage)

before adjustment. Disconnect the CNC and spindle after power-off, and turn on the power again, then, specify a frequently-used M code (M41-M44) at spindle gear (the default first gear after power-on).

② In MDI mode, specify S code at the highest speed of current gear. For example, after the first gear is selected, specify the highest speed of the first gear (parameter No. 3741) and press cycle start button.

③ Measure the output voltage SVC according to the interface definition in the above section.

④ Set the values in the following formula for parameter No. 3730

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

⑤ After the parameter is set, specify the speed analog output value of the first gear as the spindle speed of maximum voltage (parameter No. 3741), ensure that the output voltage is 10V.

⑥ Specify S0 in MDI mode.

⑦ Measure the output voltage SVC.

Set the values in the following formula for parameter No. 3731

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

Then, specify S0 again and ensure the voltage is 0V.

2. The spindle encoder is used:

① Set the parameter No. 3730 to 1000 (spindle speed analog output gain adjust data) and parameter No. 3731 to 0 (compensation value of spindle speed analog output offset voltage) before adjustment. Connect the CNC and spindle after power-off, and turn on the power again, then, specify a frequently-used M code (M41-M44) at spindle gear (the default first gear after power-on).

② In MDI mode, specify S code at the highest speed of current gear. For example, after the first gear is selected, specify the highest speed of the first gear (parameter No. 3741) and press cycle start button.

③ The actual speed is displayed in position screen, and it should be almost the same with the specified speed. If the value is too much different from the specified one, please check whether the encoder parameter setting is correct.

④ Set the values in the following formula for parameter No. 3730

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

⑤ After the parameter is set, specify the speed analog output value of the first gear as the spindle speed of maximum voltage (parameter No. 3741), ensure that the actual speed is the value set by parameter No. 3741.

⑥ Specify S0 in MDI mode.

⑦ Record the actual speed in position screen.

⑧ Input the record actual speed in parameter No. 3731

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

Relevant Parameter		
No.	**Bit**	**Meaning**
3031		Set the permitted digits in S code
3705	**#4**	When spindle function is used (spindle analog output or spindle serial output), for S command: 0: S code and SF are not output 1: S code and SF are output
3708	**#0**	Whether check spindle speed arrival signal: 0: No 1: Yes
3709	**#0**	Sampling frequency for average spindle speed 0: 4 times (usually set to 0) 1: 1 time
3730		Gain adjust data of spindle speed analog output
3731		Compensation value of spindle speed analog output offset voltage
3740		The delay time of spindle speed arrival signal
3741		The maximum spindle speed at gear 1
3742		The maximum spindle speed at gear 2
3743		The maximum spindle speed at gear 3
3744		The maximum spindle speed at gear 4
3772		Upper limit spindle speed (the first spindle)

4.9.3 Double-Spindle Control

GSK988T has two spindle interfaces for servo spindle control and analog spindle control respectively. The servo spindle, as the first spindle, can perform contour control in machining; it also can be used as analog spindle. The analog spindle, as the second spindle, provides the spindle speed control function.

Before the spindle function is used, the following setting should be made according the actual machine tool (see section 4.9.2 for relevant setting of the first spindle):

1. Set parameter No. 8133#3 (MSP) to determine whether multi-spindle function (the second spindle) is used. When the parameter is set to 0, only the first spindle (i.e. servo spindle) is used.
2. Set the maximum speed of two spindles respectively at corresponding gears (the first spindle has 4 gears, the second spindle has 2 gears); parameter No. 3811, 3812 set the maximum speed of the second spindle at the 2 gears.
3. Set the upper limit speed of two spindles (the actual speed cannot exceed the upper limit); parameter No. 3802 sets the upper limit speed of the second spindle.
4. To read the actual speed, parameters related to spindle encoder should be set as follows:
   a. the encoder line number: set by parameter No. 3803 (the second spindle)
   b. the gear ratio of encoder: set by parameter No. 3707#0, No. 3707#1)

Relevant Parameter		
No.	**Bit**	**Meaning**
3802		The upper limit speed of the second spindle
3811		Maximum speed of the second spindle at gear 1
3812		Maximum speed of the second spindle at gear 2
8133	**#3**	Whether the multi-spindle function is used: 0: No 1: Yes

CHAPTER V PARAMETER INSTRUCTION

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

Data Types	Range
(1) Bit	8 digits, 0 or 1
(2) Bit axis	
(3) Word	-99 999 999～+99 999 999
(4) Word axis	

For the (3) and (4) types, the exact data 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)

『Way of Validating』: Become valid immediately or after power-on

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

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

『Parameter Type』: Bit, bit axis, word, word axis

Note 1: The 『Value Range』 of bit type parameters is 0 or 1.
Note 2: When 『Way of Validating』 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.

(1) Bit (axis) type:

Parameter number	#7	#6	#5	#4	#3	#2	#1	#0
0000								**ABCx**

『Modification Authority』: System authority

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 ABCx The introduction of the parameter bit (axis) type is:

0: Allowed

1: Forbidden

(2) Word (axis) type:

1000	**Parameter name**

『Modification Authority』: Equipment management authority

『Way of Validating』: After power-on

『Value Range』: 0～999

Explanation information of parameter in word (axis) type

5.1 Parameters of System Setting

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

『Modification Authority』: Equipment management authority

『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.

5.2 Parameters of the Interfaces of Input and Output

0123	Baud rate of serial port (BPS)

『Modification Authority』:Equipment management

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

『Default Setting』: 115200

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

『Modification Authority』: Equipment management authority

『Default Setting』: 0000 0000

#6 OWN When NC data or the programs are input or output, whether the covered file information is displayed:

0: Yes

1: No

5.3 Parameters of Axis Control/Setting Unit

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

『Way of Validating』: 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	

『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, the same as manual reference point return.

1: P/S 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. If G28 is executed before reference point setting, P/S alarm is issued.

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

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

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

ISC	Least input unit, least command increment	Abbreviation
0	0.001mm, 0.001deg or 0.0001inch	IS-B
1	0.0001mm, 0.0001deg or 0.00001inch	IS-C

#6 RPR Whether set the minimum input unit of the rotary axis as 10 times of the minimum command increment

0: Not set it as 10 times

1: Set as it 10 times

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

『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 (MEM, 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

『Way of Validating』: 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 The machine coordinate value displays in 0～360° cycle. The stored pitch error compensation is of 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 The machine coordinate value, the relative coordinate value and the absolute coordinate value are in the linear axis, which can't display in cycle of 0～360°. The stored pitch error compensation is of the linear axis type. The cycle function and the indexing function of the rotary axis can not be used at the same time.

#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
1008						RRLx	RABx	ROAx

『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

Note1: RRLx is valid only when ROAx is 1.
Note2: The movement amount of each turn is set by parameter No.1260.

1010 Quantity of CNC controlled axes (CCA)

『Way of Validating』: After power-on

『Data Range』：0～total number

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

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

『Modification Authority』: Equipment management authority

『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 data units are

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

1: 1 ms

1020 Programming name of each axis (CAN)

『Parameter Type』: Word axis

『Value Range』: 88(X), 89(Y), 90(Z), 65(A), 66(B), 67(C)

Set the axial name of each controlled axis.

Note: The same axial name can not be set. The address used by the 2nd miscellaneous function can not be taken as the axial name.

1022 The property of each axis in the basic coordinate system

『Way of Validating』: 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

Set each controlled axis as one of three basic axes---X, Y and Z axes in the basic coordinate system, or the parallel axis which is paralleled with these axes. 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)

『Way of Validating』: After power-on

『Value Range』: 1～quantity of controlled axes

『Parameter Type』: Word axis

Set each controlled axis as the corresponding N^{th} servo axis. Generally, the setting value of the control axial number and that of the servo axial number are same. The so-called controlled axis number is to set parameter in the axis or the serial number of the signal in the axis. When the spindle is taken as the controlled axis, it is set as 5.

5.4 Parameters of the Coordinate System

	#7	#6	#5	#4	#3	#2	#1	#0
1201	WZR					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

#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 authority

『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

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

『Modification Authority』: Equipment management authority

『Value Range』: -9999 9999～9999 9999

『Parameter Type』: Word axis

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 authority

『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 authority

『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 authority

『Way of Validating』: After power-on

『Parameter Type』: Word axis

『Value Range』: 1000～9 999 999

Set the movement amount of each turn in rotary axis.

5.5 Parameters 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 authority

『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

Note: The tool stops before or after the maximum distance F/7500(mm) far away from the boundary. (F: Feedrate during reaching the boundary (Unit: mm/min))

	#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 authority

『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 and negative directions in the mechanical coordinate system in each axis stroke detection 1 in memory type. Set the outside of boundary as the restricted area to tools.

Note1: The axes specified by diameter are set by diameter value.

Note2: 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.

Note3: If parameter LMS (No. 1300#2) is "1", and the stroke limit switching signal in memory type EXLM is also "1", the restricted area is invalid set by the parameter. Parameter No.1326 and No.1327 set the restricted area.

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

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

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Default Setting』: NO.1322 is 99 999 999, NO.1323 is－99 999 999

『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 each axis stroke detection 2 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 3 in memory type (NC3)

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Default Setting』: No.1324 is 99 999 999, No.1325 is －99 999 999

『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 each axis stroke detection 3 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 (NC12)

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Default Setting』: NO.1326 is 99 999 999, NO.1327 is －99 999 999.

『Value Range』: －99 999 999～99 999 999

Respectively set the positive and negative boundary coordinate values of each axis stroke detection 1 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.

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

5.6 Parameters of the Feedrate

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

『Modification Authority』: Equipment management authority

『Default Setting』: 0000 0000

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

0: Invalid (JOG speed)

1: Valid

#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 authority

『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							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)

#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 authority

『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: According to the following table

SETTING UNITS	UNIT	IS-B	IS-C
Input in metric system	mm/min	0.001～60000	0.001～24000
Input in inch system	inch/min	0.00001～2400	0.00001～960
Rotary axis	deg/min	1～60000	1～24000

1410	Dry run speed (DRR)

『Parameter Type』: Word type

『Value Range』:

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

Set the speed during dry run.

1411	Feedrate in auto mode after power on (IFV)

『Parameter Type』: Word type

『Value Range』:

SETTING UNITS	DATA UNITS	VALID RANGE	DEFAULT SETTING
Machine in metric system	1 mm/min	6～32767	1000
Machine in inch system	0.1 inch/min		

It doesn't require changing the cutting speed in the machine during the processing. And the cutting feedrate can be set by the parameter, 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)**

『Parameter Type』: Word axis

『Value Range』:

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

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	DATA 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	30～6000	30～4800	
Rotary axis	1 deg/min	30～15000	30～12000	

Set the speed when the rapid movement override of each axis is 0.

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

『Parameter Type』: Word type

『Value Range』:

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

Set the maximum cutting feedrate for all axes.

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

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6～32767		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	DATA UNIT	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Metric machine	1 mm/min	0, 30～60000		8000
Inch machine	0.1 inch/min			
Rotary axis	1 deg/min			

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 authority

『Parameter Type』: Word axis

『Value Range』:

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

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

5.7 Parameters of Control of Acceleration/Deceleration

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

『Modification Authority』: Equipment management authority

『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				JGLx				CTLx

『Modification Authority』: Equipment management authority

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 CTLx Acceleration/deceleration of cutting and feeding (include feeding during dry run)

0: Acceleration/deceleration in index type

1: Acceleration/deceleration in linear type after interpolation

#4 JGLx The acceleration/deceleration during JOG feeding

0: Acceleration/deceleration in index type

1: Acceleration/deceleration in linear type after interpolation

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

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』: 0～4000 ms

『Default Setting』: 100

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

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

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』: 0～4000 ms

『Default Setting』: 100

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

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.

1623	FL speed of acceleration/deceleration in index type of each axis cutting and feeding (FLC)

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

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

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

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

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』: 0～4000ms

『Default Setting』: 100

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

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

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

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

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

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

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

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』: 0～4000ms

『Default Setting』: 100

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

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

『Modification Authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	DATA 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, 6～6000	0, 6～4800	30

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

5.8 Parameter of Servo and Backlash Compensation

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

『Default Setting』: 1000 0000

#4 RBK: Cutting feeding and rapid movement are respectively compensated with backlash

0: No compensate

1: Compensate

#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 parameter #1853 and #1800.6.

1: Pulse frequency output based on the acceleration/deceleration characteristics

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

『Way of Validating』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 ABP Selecting pulse drive modes

0: Pulse +direction mode

1: AB phases pulse mode

#2 POD Selecting output directions of each axis pulse

0: Not reversed

1: Reversed

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

『Way of Validating』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 APRx The position direction of the absolute position detector during using the absolute position encoder

0: Not negate

1: Negate

#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)

	#7	#6	#5	#4	#3	#2	#1	#0
1816		DM3x	DM2x	DM1x				ISAx

『Way of Validating』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0001 0001

#0 ISAx Servo alarm signal

0: High level of alarm signal is valid

1: Low level of alarm signal is valid

#4-#6 DM1x-DM3x: Set the detection multiply ratio of each axis (DMR)

SETTING VALUE			DETECTION MULTIPLY RATIO (DMR)
DM3x	DM2x	DM1x	
0	0	0	1/2
0	0	1	1
0	1	0	3/2
0	1	1	2
1	0	0	5/2
1	0	1	3
1	1	0	7/2
1	1	1	4

1820	Command multiply ratio of each axis (CMR)

『Parameter Type』: Word axis

『Value Range』:

COMMAND MULTIPLY RATIO (CMR)	VALID RANGE OF VALUE SET BY NO.1820	DEFAULT SETTING
1/2～1/27	102～127	2
1 ～ 48	2～96	

Set the command multiply ratio (CMR) of each axis.

1. When the command multiply ratio (CMR)is 1/2～1/27, the setting value = 1 / CMR＋100;

2. When the command multiply ratio (CMR)is 1～48, the setting value = 2×CMR.

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:

			MINIMUM SETTING UNITS	MINIMUM MOVEMENT UNITS
IS－B	Metric machine	Input in metric system	0.001mm (specified by the diameter)	0.0005mm
			0.001mm (specified by the radius)	0.001mm
		Input in inch system	0.0001 inch (specified by the diameter)	0.0005mm
			0.0001 inch (specified by the radius)	0.001mm
	Inch machine	Input in metric system	0.001mm (specified by the diameter)	0.00005 inch
			0.001mm (specified by the radius)	0.0001 inch
		Input in inch system	0.0001 inch (specified by the diameter)	0.00005 inch
			0.0001 inch (specified by the radius)	0.0001 inch
	Rotary axis		0.001deg	0.001deg

			MINIMUM SETTING UNITS	MINIMUM SETTING UNITS
IS－C	Machine in metric system	Input in metric system	0.0001mm (specified by the diameter)	0.00005mm
			0.0001mm (specified by the radius)	0.0001mm
		Input in inch system	0.00001 inch (specified by the diameter)	0.00005mm
			0.00001 inch (specified by the radius)	0.0001mm
	Machine in	Input in	0.0001mm (specified	0.000005 inch

	inch system	metric system	by the diameter)	
			0.0001mm (specified by the radius)	0.00001 inch
		Input in inch system	0.00001 inch (specified by the diameter)	0.000005 inch
			0.00001 inch (specified by the radius)	0.00001 inch
	Rotary axis		0.0001deg	0.0001deg

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

『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.

1852 **Backlash compensation value during each axis rapid movement (BCVR)**

『Parameter Type』: Word axis

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

『Default Setting』: 0

Set the backlash compensation value during each axis rapid movement. It is valid when parameter NO.1800#4(RBK) is set as 1. It can change the backlash compensation value based on the cutting feedrate/rapid movement speed to process in higher precision.

Note 1: Manually continuous feeding (JOG) is taken as cutting feed.

Note 2: After connecting power supply and before the reference point return completes at the first time, it doesn't compensate the backlash in cutting feed/rapid movement. No matter the compensation value is the cutting feed or the rapid movement, it should be compensated based on parameter NO.1851.

Note 3: When parameter NO.1800#4(RBK) is set as 1, parameter NO.1851 is the backlash compensation value of cutting feed, parameter NO.1852 is the backlash compensation value of rapid movement. When parameter NO.1800#4(RBK) is set as 0, parameter NO.1851 is the backlash compensation value of cutting feed/rapid movement.

	#7	#6	#5	#4	#3	#2	#1	#0
1853				**CPF5**	**CPF4**	**CPF3**	**CPF2**	**CPF1**

『Default Setting』: 0000 0111

CPF1～CPF5:Settting value of the backlash compensation pulse frequency (in BCD code)

Setting frequency= (setting value +1) Kpps

CPF5	CPF4	CPF3	CPF2	CPF1	SETTING FREQUENCY (Kpps)
0	0	0	0	0	1
0	0	0	0	1	2
0	0	0	1	0	3

0	0	0	1	1	4
0	0	1	0	0	5
0	0	1	0	1	6
0	0	1	1	0	7
0	0	1	1	1	8
0	1	0	0	0	9
0	1	0	0	1	10
0	1	0	1	0	11
0	1	0	1	1	12
0	1	1	0	0	13
0	1	1	0	1	14
0	1	1	1	0	15
0	1	1	1	1	16
1	0	0	0	0	17
1	0	0	0	1	18
1	0	0	1	0	19
1	0	0	1	1	20
1	0	1	0	0	21
1	0	1	0	1	22
1	0	1	1	0	23
1	0	1	1	1	24
1	1	0	0	0	25
1	1	0	0	1	26
1	1	0	1	0	27
1	1	0	1	1	28
1	1	1	0	0	29
1	1	1	0	1	30
1	1	1	1	0	31
1	1	1	1	1	32

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

『Parameter Type』: Word axis

『Value Range』: 0～100 ms

『Default Setting』: 40

Set each axis backlash acceleration/deceleration valid time constant.

5.9 Parameter of Input/Output

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

『Default Setting』: 1000 0000

#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					

『Default Setting』: 0010 0000

#5 OTH Overtravel limit signal

0: Check

1: Not check

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

『Default Setting』: 0000 0000

#0 GDC Deceleration signal of the reference point return

0: Use X signal

1: Use G196 (X signal is invalid)

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

『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)

『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 finish signals (FIN)of M, T and S

『Parameter Type』:Word type

『Default Setting』: 16

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

Note: Time is set by 8ms, if the setting value is not the multiple of 8, it should be carried into the multiple of 8.

3017	Output time of the resetting signal (RST)

『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.

3030	Allowable digits of M code (MCB)

『Value Range』: 2～8

『Default Setting』: 2

Set the allowable digits of M code.

3031	Allowable digits of S code (SCB)

『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)

『Value Range』: 2～8

『Default Setting』: 4

Set the allowable digits of T code.

5.10 Parameter of Display and Editing

	#7	#6	#5	#4	#3	#2	#1	#0
3101				BGD				

『Modification Authority』: Equipment management authority

『Default Setting』: 0000 0000

#4 BGD Background editing selects the programs selected at the foreground

0: Editable

1: Ineditable

	#7	#6	#5	#4	#3	#2	#1	#0
3102					CHI			

『Way of Validating』: After power-on

『Default Setting』: 0000 1000

#3 CHI Display language

0: English

1: Chinese

Set the selected language for display.

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC	DAL	DRC	DRL				MCN

『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.

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				**SOR**	**REV**	**DNC**		

『Modification Authority』: Equipment management authority

『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

#4 SOR Display orders of program directory

0: Based on the time sequence

1: Based on the program numbers

	#7	#6	#5	#4	#3	#2	#1	#0
3110						**AHC**		

『Modification Authority』: Equipment management authority

『Default Setting』: 0000 0000

#2 AHC Whether the alarm resume can be cleared by softkeys

0: Yes

1: No

	#7	#6	#5	#4	#3	#2	#1	#0
3111	**NPA**							

『Modification Authority』: Equipment management authority

『Default Setting』: 0000 0000

#7 AHC Whether switch to alarm/information screen when alarm occurs or information is input:

0: Yes

1: No

	#7	#6	#5	#4	#3	#2	#1	#0
3114								IPC

『Modification Authority』: Equipment management authority

『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
3202			CPD					

『Modification Authority』: Equipment management authority

『Default Setting』: 0000 0000

#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 authority

『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 authority

『Default Setting』: 0000 0000

#0 MPD When the subprogram is executed, whether display the main program number

0: Not display

1: Display

3216	**the increment value (INC) during the serial number being auto inserted**

『Modification Authority』:Equipment management authority

『Value Range』: 0～9999

『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.

5.11 Parameter of Programming

	#7	#6	#5	#4	#3	#2	#1	#0
3401						**NCK**		**DPI**

『Modification Authority』: Equipment management authority

『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

	#7	#6	#5	#4	#3	#2	#1	#0
3402	**G23**	**CLR**		**FPM**				**G01**

『Modification Authority』: Equipment management authority

『Default Setting』: 0001 0000

#0 G01 Mode during connecting the power supply

0: G00 mode (orientation)

1: G01 mode (linear interpolation)

#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 authority

『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, 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 authority

『Default Setting』: 0000 0000

#4 M30 During auto running, process M30 command

0: M30 auto searches the program header when it is sent to the machine side. Therefore, the program is executed from the beginning when the resetting isn't executed or return to the resetting and the finish signal FIN returns.

1: M30 is sent to the machine side, but it doesn't return to the beginning of the program.

#5 M02 During auto running, process M02 command

0: M02 auto searches the program header when it is sent to the machine side. The program is executed from the beginning when resetting isn't executed or resetting returns and finish signal FIN returns,.

1: M02 is sent to the machine side, but it 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)

#7 M3B The quantity of M codes which can be commanded in one block

0: One

1: Maximum three

3410	Circular radius allowable error(CRE)

『Modification Authority』: Equipment management authority

『Value Range』: 0～9999 9999

『Default Setting』: 0

SETTING UNITS	IS－B	IS－C	UNITS
input in mm	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

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.

5.12 Parameters of the Screw Pitch Error Compensation

3620	Screw pitch error compensation number in each axis reference point (NPR)

『Way of Validating』: 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)

『Way of Validating』: 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)

『Way of Validating』: 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.

The parameter setting value should be greater than that of parameter NO.3620.

3623	Each axis screw pitch error compensation override (PCM)

『Way of Validating』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～100

『Default Setting』: 0

Set the override of each axis screw pitch error compensation.

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)

『Way of Validating』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～100

『Default Setting』: 0～99 999 999

『Default Setting』: 0

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
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 1: The unit of the screw pitch compensation value is same as that of the detection.
Note 2: 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.

	#7	#6	#5	#4	#3	#2	#1	#0
3628				NPF5	NPF4	NPF3	NPF2	NPF1

『Default Setting』: 0000 0111

#0~#4 NPF1～NPF5 Setting value of the screw pitch compensation pulse frequency (in BCD code).

Setting frequency= (setting value +1) Kpps

NPF5	NPF4	NPF3	NPF2	NPF1	Setting frequency (Kpps)
0	0	0	0	0	1
0	0	0	0	1	2
0	0	0	1	0	3
0	0	0	1	1	4
0	0	1	0	0	5
0	0	1	0	1	6
0	0	1	1	0	7
0	0	1	1	1	8
0	1	0	0	0	9
0	1	0	0	1	10
0	1	0	1	0	11
0	1	0	1	1	12
0	1	1	0	0	13
0	1	1	0	1	14
0	1	1	1	0	15

0	1	1	1	1	16
1	0	0	0	0	17
1	0	0	0	1	18
1	0	0	1	0	19
1	0	0	1	1	20
1	0	1	0	0	21
1	0	1	0	1	22
1	0	1	1	0	23
1	0	1	1	1	24
1	1	0	0	0	25
1	1	0	0	1	26
1	1	0	1	0	27
1	1	0	1	1	28
1	1	1	0	0	29
1	1	1	0	1	30
1	1	1	1	0	31
1	1	1	1	1	32

5.13 Parameters of the Spindle Control

	#7	#6	#5	#4	#3	#2	#1	#0
3705				**EVS**				

『Modification Authority』: Equipment management authority

『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							**PG2**	**PG1**

『Default Setting』: 0000 0000

#0, #1 PG2 and PG1 Gear ratio between the spindle and the position encoder.

Gear ratio=spindle speed/position encoder speed

Gear ratio	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

	#7	#6	#5	#4	#3	#2	#1	#0
3707							**P22**	**P21**

『Default Setting』: 0000 0000

#0, #1 P22 and P21 Gear ratio between the spindle and the second position encoder.

Gear ratio= spindle speed/position encoder speed

Gear ratio	P22	P21
×1	0	0
×2	0	1
×4	1	0
×8	1	1

Note: The parameter is valid only when multi-spindle control.

	#7	#6	#5	#4	#3	#2	#1	#0
3708		**TSO**					**SAT**	**SAR**

『Modification Authority』: Equipment management authority

『Default Setting』: 0000 0000

#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 continually executed, the spindle speed reaching signal isn't checked in the thread cutting block after the 2nd block.

#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 authority

『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.)

3730	**Increment adjustment data of the spindle speed analog output (AGS)**

『Value Range』: 700～1250

『Default Setting』: 1000

『Data unit』: 0.1%

Set the increment adjustment data 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:

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

(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)**

『Value Range』: -1024～+1024

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)**

『Value Range』: 0～255ms

『Default Setting』: 6000

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)**

『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)

『Default Setting』: 0

『Value Range』: 0, 1～quantity of the controlled axes

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)

『Value Range』: 0～32767r/min

『Default Setting』: 0

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)

『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.

3. When PLC controls the spindle speed, the parameter is invalid and the spindle speed isn't limited by the maximum speed.

4. When multi-spindle control, the maximum speed of each spindle is set through the following parameters:

The maximum speed of the 1st spindle is set by parameter NO.3772.

The maximum speed of the 2nd spindle is set by parameter NO.3802.

3773	Quantity of the spindle encoder pulses (CNT)

『Way of Validating』: After power-on

『Value Range』: 100～9999

『Default Setting』: 1024

The parameter sets the quantity of the spindle encoder pulses.

3802	Maximum speed of the 2nd spindle (MSS2)

『Value Range』: 0～32767r/min

『Default Setting』: 6000

The parameter sets the maximum speed of the 2nd spindle. 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 multi-spindle controls, the parameter is valid.
2. When the constant surface speed controls, no matter whether G96 or G97 is commanded, the spindle speed is limited by the maximum speed.
3. When the setting value is 0, parameter NO.3772 is valid (the maximum speed of the 1st spindle). When parameter NO.3772 is 0, the spindle speed is not limited.
4. When PLC controls the spindle speed, the parameter is invalid and the spindle speed isn't limited by the maximum speed.

3803	Quantity of the 2nd spindle encoder pulses (CNT2)

『Way of Validating』: After power-on

『Default Setting』: 1024

『Span』: 100～9999

The parameter sets the quantity of the 2nd spindle encoder pulses.

3811	Spindle maximum speed of the 2nd spindle gear 1 (M2G1)

3812	Spindle maximum speed of the 2nd spindle gear 2 (M2G2)

『Default Setting』: 6000

『Value Range』: 0～32767r/min

The parameter sets the maximum speed of each gear in the 2nd spindle.

Note:
It is for multi-spindle control.

5.14 Parameters of the Tool Compensation

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO		EVR				

『Modification Authority』: Equipment management authority

『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 authority

『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 authority

『Default Setting』: 0000 0000

#2 CCN In the tool nose radius compensation mode, when the auto reference point return (G28) is commanded,

0: The compensation vector of the tool nose radius is canceled when the tool nose traverses to the intermediate point.

1: The compensation vector of the tool nose radius isn't canceled when the tool nose traverses to the intermediate point. 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

	#7	#6	#5	#4	#3	#2	#1	#0
5004							ORC	

『Modification Authority』: Equipment management authority

『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

	#7	#6	#5	#4	#3	#2	#1	#0
5005						PRC		

『Modification Authority』: Equipment management authority

『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

	#7	#6	#5	#4	#3	#2	#1	#0
5006							TGC	OIM

『Modification Authority』: Equipment management authority

『Way of Validating』: 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 authority

『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

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 authority

『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 authority

『Default Setting』: 10

『Value Range』:

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

SETTING RANGE	IS-B	IS-C
Input in metric system	0～9 999 999	0～99 999 999
Input in inch system		

The parameter sets the maximum value of the tool wearing compensation value.

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.

5.15 Parameters of the 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

5.15.1 Parameters of the Drilling Canned Cycle

	#7	#6	#5	#4	#3	#2	#1	#0
5102							MRC	

『Modification Authority』:Equipment management authority

『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 retraction amount 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 authority

『Default Setting』:0000 0000

#2 FCK : In combined canned cycles (G71, G72 and G73), the processing appearance is

0: Not checked

1: Checked

5110	M code locking C axis in the canned cycle of drilling holes (CMD)

『Modification Authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: 0～99

Set M code , which can lock C axis, during the canned cycle of drilling holes.

5.15.2 Parameters of the Thread Cutting Cycle

5130	Chamfering value of the thread cutting cycle (G76, G92)(THD)

『Modification Authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: 0～99× (0.1 screw pitch)

The parameter sets the beveling value of G76 and G92 thread cutting cycle.

5.15.3 Parameters of the Combined Canned Cycle

5132	Cut-in value of the combined canned cycle G71 and G72 (THC)

『Modification Authority』:Equipment management authority

『Default Setting』: 1000

『Value Range』: 0～99 999 999

Set the cut-in value of G71 and G72 combined canned cycle.

	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

5133 **Retraction amount of G71 and G72 combined canned cycle (MCE)**

『Modification Authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: 0～99 999 999

Set the retraction amount of G71 and G72 combined canned cycle.

5135 **Retraction amount of G73 combined canned cycle along X axis direction (G73XE)**

5136 **Retraction amount of G73 combined canned cycle along Z axis direction (G73ZE)**

『Modification Authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: -99 999 999～99 999 999

Set the retraction amount 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 authority

『Default Setting』: 1

『Value Range』: 1～99 999 999

Set the partition times of G73 combined canned cycle.

5139 **Retraction amount of G74 and G75 combined canned cycles (G74G75R)**

『Modification Authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: 0～99 999 999

Set the reversal value of G74 and G75 combined canned cycle.

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

5140 **Minimum cut-in value of G76 combined canned cycle (G76MID)**

『Modification Authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: 0～99 999 999

Set the minimum cut-in value of G76 combined canned cycle.

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

5141	Finishing allowance of G76 combined canned cycle (G76FA)

『Modification Authority』:Equipment management authority

『Default Setting』: 500

『Value Range』: 1～99 999 999

Set the finishing allowance of G76 combined canned cycle.

5142	Finishing cycle times of G76 combined canned cycle (G76FC)

『Modification Authority』:Equipment management authority

『Default Setting』: 1

『Value Range』: 1～99

Set the finishing cycle times of G76 combined canned cycle.

5143	Tool nose angle of G76 combined canned cycle (G76TNA)

『Modification Authority』:Equipment management authority

『Default Setting』: 60

『Value Range』: 0～99 (deg)

Set the tool nose angle of G76 combined canned cycle.

5.16 Parameters of the Rigid Tapping

	#7	#6	#5	#4	#3	#2	#1	#0
5200		FHD		DOV		CRG		G84

『Modification Authority』:Equipment management authority

『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

0: Invalid

1: Valid, override value is set by parameter 5211

#6 FHD Feed pause and single block running in rigid tapping is:

0: Forbidden

1: Allowed

	#7	#6	#5	#4	#3	#2	#1	#0
5201	TXZ	TDK				TDR		

『Modification Authority』:Equipment management authority

『Default Setting』: 0000 0000

#2 TDR: The time constant of the rigid tapping cutting and that of the tapping run-out:

0: Same

1: Different

#6 TDK: Specify K in tapping command

0: Take it as the cycle times

1: Ignore

#7 TXZ: Non-tapping axis is taken as the orientation in tapping command

0: Allow to use

1: Alarm

5210	M code commanding the rigid tapping (RTMC)

『Modification Authority』:Equipment management authority

『Default Setting』: 29

『Value Range』: 0～255

M code is set to specify the rigid tapping method. When it is set as 0, CNC takes it as M29.

5211	Speed override value in tapping run-out (RTOV)

『Modification Authority』:Equipment management authority

『Default Setting』: 120

『Data unit』: 1% or 10%

『Value Range』: 0～200

Set the override value during the rigid tapping run-out, and it is valid only when parameter DOV (NO.5200 BIT4) is set as 1.

5241	Spindle maximum speed in rigid tapping (RTMS)

『Modification Authority』:Equipment management authority

『Default Setting』: 1000

『Value Range』: 0～9999

Set the spindle maximum speed in rigid tapping.

5261	Time constant of linear acceleration and deceleration in rigid tapping (RTLT)

『Modification Authority』:Equipment management authority

『Default Setting』: 200

『Value Range』: 0～4000ms

During the rigid tapping, the time constant of linear acceleration or deceleration of the spindle and the tapping axis is the time (parameter NO.5241) of the spindle maximum speed when the spindle reaches the rigid tapping. The actual time is the ratio between the specified spindle speed and the maximum speed multiplies by the parameter.

5271	Linear acceleration or deceleration time constant in rigid tapping run-out (RTET)

『Modification Authority』:Equipment management authority

『Default Setting』: 200

『Value Range』: 0～4000ms

Set the time constant of linear acceleration or deceleration of the spindle and the tapping axis during the rigid tapping run-out. The parameter is valid only when parameter TDR (NO.5201 BIT2) is set as 1.

5.17 Parameters of the Polar Coordinates Interpolation

	#7	#6	#5	#4	#3	#2	#1	#0
5450							AFC	

『Modification Authority』:Equipment management authority

『Default Setting』: 0000 0000

#0 AFC: Whether use the auto override and the auto speed in the polar coordinates interpolation mode.

0: Not use

1: Use

Note:In the polar coordinates interpolation mode, the more closely the tool is near to the work piece center, the bigger the speed vector of the revolving 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 revolving axis doesn't exceed the maximum cutting feedrate.

5460	Specify the polar coordinates interpolation axis (linear axis) (LAI)

5461	Specify the polar coordinates interpolation axis (revolving axis) (RAI)

『Value Range』: 1～quantity of the controlled axes

『Default Setting』: NO.5460 is 0； NO.5461 is 5

Set the controlled axis numbers of the linear axis and the revolving axis for polar coordinates interpolation.

5462	Maximum cutting feedrate of the polar coordinates interpolation (MFI)

『Default Setting』: 8000

『Value Range』:

	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
Rotary axis	0, 6～24 000	0, 6～10 000	deg/min

Set the valid maximum feedrate of the polar coordinates 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 coordinates interpolation is limited by the maximum cutting feedrate (parameter NO.1422) value.

5463	Allowable auto override percentage in polar coordinates interpolation (API)

『Value Range』: 1～quantity of the controlled axes

『Default Setting』: 0

『Value Range』: 0～100 (%)

When the polar coordinates interpolation is set, the percentages of the auto override are allowed

to limit the cutting feedrate of the revolving axis.

The allowable speed of the revolving axis = Maximum cutting feedrate X override percentage

In polar coordinates interpolation, the more closely the tool is near to the work piece center, the bigger the speed vector of the revolving 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 revolving axis/the speed vector of the revolving 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.

5.18 Parameters of the User Macro Program

	#7	#6	#5	#4	#3	#2	#1	#0
6000			**SBM**					**G67**

『Modification Authority』:Equipment management authority

『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 authority

『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 authority

『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 -180.0～0～180.0

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

5.19 Parameters of the Skip Function

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF						SK0	

『Default Setting』: 0000 0000

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”

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 authority

『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

『Way of Validating』: 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)**

『Data setting』:

SETTIN UNIT	DATA UNIT	VALID RANGE (IS-B/ IS-C)	DEFAULT
Metric	1mm/min	6～15000	1000
Inch	0.1inch/min		

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 authority

『Default Setting』: 1000

『Data range』: 1～99999999

These two parameters set the γ value in tool compensation function in sequence.

Note: The value is set in radius no matter diameter or radius programming is specified.

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 authority

『Data 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

5.20 Parameters of the Graphic Display

	#7	#6	#5	#4	#3	#2	#1	#0
6500					DPA			

『Modification Authority』: Equipment management authority

『Default Setting』: 0000 0000

#3 DPA In the graphic display interface, the current position displays

0: Display the actual position including the tool compensation and offset

1: Display the programming position excluding the tool compensation and offset

5.21 Parameter of Run Hour and Parts Count Display

	#7	#6	#5	#4	#3	#2	#1	#0
6700							PRT	PCM

『Modification Authority』: Equipment management authority

『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)

『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, neither.

6713	Quantity of the required parts (RPM)

『Value Range』: 0～9 999

『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, if the quantity is 0, it is regarded as infinitely great, not output to PRTSF.

5.22 Parameter of MPG Feed

	#7	#6	#5	#4	#3	#2	#1	#0
7100				HPF				JHD

『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 doesn't 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

『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

7110	Quantity of MPG (NMP)

『Value Range』: 0～2

『Default Setting』: 1

Set the quantity of MPG.

7113	MPG feeding override M (MFM)

『Value Range』: 1～127

『Default Setting』: 100

Set the override when MPG feeding movement value selection signals MP1=0, MP2=1.

7114	MPG feeding override N (MFN)

『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 feeding (APM)

『Value Range』: 0～99999999

『Default Setting』: 10000

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 unit the remaining pulse is completed.

5.23 Parameter of PLC Axis Control

	#7	#6	#5	#4	#3	#2	#1	#0
8001			NCC		RDE	OVE		MLE

『Default Setting』: 0000 0000

#0 MLE Whether the locking machine signal MLK of PLC controlled 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 controlled axes (the controlled 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.

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10		DWE	RPD

『Default Setting』: 0000 0000

#0 RPD The rapid movement speed of PLC controlled axis

0: Feedrate set by parameter No.1420

1: In axis control command, feedrate set by feedrate data

#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

PR2	PR1	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
8003								PIM

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 PIM If PLC controlled axis is linear axis, the control commands are

0: Affected by inch system/metric system

1: Not affected by inch system/metric system

	#7	#6	#5	#4	#3	#2	#1	#0
8004	NDI	NCI	DSL			JFM	NMT	CMV

『Default Setting』: 0000 0000

#0 CMV According to the commands sent by CNC, PLC sends the axis control command after moving along the axis and before receiving the command signal of the miscellaneous function.

0: P/S No.130 alarms

1: The axis is processed as one PLC axis and is executed the set movement.

#1 NMT: When PLC is processing one control command of some axis, CNC sends another command to command the axis and PLC controlled axis does not move.

0: P/S No.133 alarms

1: Not alarm

#2 JFM Feedrate units of continuous feeding (06h) of PLC controlled 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 deceleration of the axes controlled by PLC, in-position check is

0: Executed

1: Not executed

#7 NDI: When PLC controlled axis selects the diameter programming, under PLC axis control

0: The radius programming specifies the movement distance and the feedrate

1: The diameter programming specifies the movement distance and the feedrate

	#7	#6	#5	#4	#3	#2	#1	#0
8005							**CDI**	

『Default Setting』: 0000 0000

#1 CDI When PLC controlled axis selects the diameter programming, under PLC axis control

0: Radius programming specifies the movement distance and the feedrate

1: The diameter programming specifies the movement distance and the radius programming specifies the feedrate

8010	**Selecting each axis DI/DO group controlled by PLC (PAS)**

『Parameter Type』: Word type

『Default Setting』: 0

『Value Range』: 0～4

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 (PAMS)**

『Parameter Type』: Word type

『Default Setting』: 6

『Value Range』:

INCREMENT SYSTEM	**DATA UNITS**	**VALID DATA RANGE**	
		IS-B	**IS-C**
Machine in metric system	1mm/min	6~15000	6~12000
Machine in inch system	0.1inch/min	6~6000	6~4800
Rotary axis	1deg/min	6~15000	6~12000

Set the maximum feedrate of feeding/per revolution controlled by PLC axis.

8028	For each PLC controlled axis, the linear acceleration or deceleration time constant specified by speed command during JOG feeding (PALT)

『Parameter Type』: Word axis

『Default Setting』: 200

『Value Range』: 0~3000ms

Specify the linear acceleration or deceleration time constant during JOG feeding

Note: If it is specified as "0", the system doesn't control the acceleration/deceleration.

5.24 Parameters of the Basic Function

8130	Total quantity of the controlled axes (TCA)

『Way of Validating』: After power-on

『Default Setting』：2

『Data Range』：2～5

Set the total quantity of the axes controlled by CNC system.

	#7	#6	#5	#4	#3	#2	#1	#0
8131								HPG

『Way of Validating』: 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								TLF

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 TLF : Whether use the tool work life management function

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
8133					MSP	SCS	AXC	SSC

『Way of Validating』: 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

#1 AXC Whether the use spindle orientation function

0: Not use

1: Use

#2 SCS Whether use CS outline control function

0: Not use

1: Use

#3 MSP Whether use the multi-spindle control function

0: Not use

1: Use

5.25 Parameters of GSKLink Communication Function

	#7	#6	#5	#4	#3	#2	#1	#0
9000								**ACAN**

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 ACAN: Whether the system servo communication function is valid

0: Invalid

1: Valid

9010	**Baud rate of the system servo communication (ABPS)**

『Way of Validating』: After power-on

『Default Setting』: 500 (kbps)

『Data Range』; 500, 600, 800 or 1000 (kbps)

ABPS The parameter sets the baud rate of the system servo communication

9011	**Slave number corresponding to each axis during servo communication (SIDx)**

『Way of Validating』: After power-on

『Value Range』: 0~5

『Default Setting』: 0

SIDx The parameter sets the slave number corresponding to each axis during servo communication.

Note: "0" represents the axis doesn't connect with the servo slave. "1~5" represent the servo slave number corresponding to each axis.

9012	**Slave number corresponding to the extended servo spindle communication**

『Way of Validating』: After power-on

『Default Setting』: 0

『Value Range』: 0~5

SIDS: During the servo spindle communication, the parameter sets the corresponding slave numbers out of the range of the controlled axes total quantity.

Note:"0" represents the axis doesn't connect with the servo slave. "1~5" represent the analog spindle slave number corresponding to the axis.

CHAPTER VI STANDARD PLC FUNCTION CONFIGURATION

6.1 Standard Panel on the Machine Tool

Fig. 6-1-1 Standard layout of operation panel

6.2 Addresses X, Y Definition

Caution:

The general I/O signal (except those signals marked for fixed addresses) in GSK988T CNC system is defined by the embedded PLC (ladder diagram) program. When this CNC system is installed, the exact I/O functions are determined by the machine tool builder. Please refer to the manual from machine tool builder for details.

Pay attention that in this chapter, the functions of general I/O signal (i.e. X,Y addresses) are just described for GSK988T standard PLC program.

6.2.1 General I/O Interface on Machine Tool

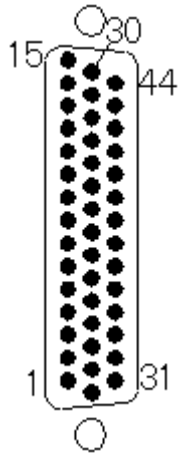


Fig 6-2-1 CN61 (male)input

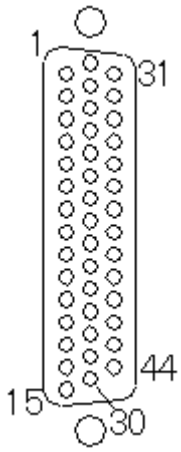


Fig. 6-2-2 CN62 (female) output

DB Pin	PLC address	Function defined by standard PLC address		Remark
CN61.1	X0.0	SAGT	Protection door detection signal	
CN61.2	X0.1		Reserved	
CN61.3	X0.2	DIQP	Chuck input signal	
CN61.4	X0.3	DEC1	The 1st axis deceleration signal	Fixed address
CN61.5	X0.4	DITW	Tailstock control signal	
CN61.6	X0.5	ESP	Emergency stop input signal	Fixed address
CN61.7	X0.6	PRES	Pressure detection signal	
CN61.8	X0.7	T05	Tool position signal 5/ tool post pre-indexing signal (Yantai AK31)/Sensor E (Liuxin Tool Post)	
CN61.9	X1.0	T06	Tool position signal 6/ tool post pre-indexing signal (Yantai AK31)/Sensor F (Liuxin Tool Post)	

DB Pin	PLC address	Function defined by standard PLC address		Remark
CN61.10	X1.1	T07	Tool position signal 7/ tool post overheat signal (Yantai AK31)	
CN61.11	X1.2	T08	Tool position signal 8	
CN61.12	X1.3	DEC3	The 3rd axis deceleration signal	Fixed address
CN61.13	X1.4		Reserved	
CN61.14	X1.5	M41I	The 1st gear stage in-position	
CN61.15	X1.6	M42I	The 2nd gear stage in-position	
CN61.16	X1.7	T01	Tool position signal 1/T1 (Yantai AK31)/Sensor A (Liuxin Tool Post)	
CN61.29	X2.0	T02	Tool position signal 2/T2 (Yantai AK31)/ Sensor B (Liuxin Tool Post) Sensor A (Liuxin Tool Post)	
CN61.30	X2.1	T03	Tool position signal 3/T3 (Yantai AK31)/Sensor C (Liuxin Tool Post)	
CN61.31	X2.2	T04	Tool position signal 4/T4 (Yantai AK31)/Sensor D (Liuxin Tool Post)	
CN61.32	X2.3	DEC2	The 2nd axis deceleration signal	Fixed address
CN61.33	X2.4	DEC4	The 4th deceleration signal	Fixed address
CN61.34	X2.5	DEC5	The 5th deceleration signal	Fixed address
CN61.35	X2.6	TCP	Tool post lock signal Tool post proximity switch signal (Yantai AK31)	
CN61.36	X2.7	COIN	Spindle orientation completed signal	
CN61.37	X3.0	LMI1+	The 1st axis + side overtravel signal	
CN61.38	X3.1	LMI2+	The 2nd axis + side overtravel signal	
CN61.39	X3.2	LMI3+	The 3rd axis + side overtravel signal	
CN61.40	X3.3	WQPJ	Chuck in-position signal (outer chuck clamping and inner chuck unclamping)	
CN61.41	X3.4	NQPJ	Chuck in-position signal (inner chuck clamping and outer chuck unclamping)	
CN61.42	X3.5	SKIP	G31 skip signal	Fixed address
CN61.43	X3.6	G36	G36 skip signal	Fixed address
CN61.44	X3.7	G37	G37 skip signal	Fixed address

DB Pin	PLC address	Function defined by standard PLC address		Remark
CN61.17	X4.0	LMI1-	The 1st axis – direction overtravel signal	
CN61.18	X4.1	LMI2-	The 2nd axis – direction overtravel signal	
CN61.19	X4.2	LMI3-	The 3rd axis – direction overtravel signal	
CN61.20	X4.3	LMI4+	The 4th axis + direction overtravel signal	
CN61.25	X4.4	LMI4-	The 4th axis - direction overtravel signal	
CN61.26	X4.5	LMI5+	The 5th axis + direction overtravel signal	
CN61.27	X4.6	LMI5-	The 5th axis - direction overtravel signal	
CN61.28	X4.7		Reserved	
CN61.21～CN61.24	0V			
CN62.1	Y0.0	M08	Cooling output signal	
CN62.2	Y0.1	M32	Lubrication output signal	
CN62.3	Y0.2		Reserved	
CN62.4	Y0.3	M03	Spindle CCW signal	
CN62.5	Y0.4	M04	Spindle CW signal	
CN62.6	Y0.5	M05	Spindle stop signal	
CN62.7	Y0.6		Reserved	
CN62.8	Y0.7	SPZD	Spindle braking output signal	
CN62.9	Y1.0	M41	Spindle gear 1 output signal	
CN62.10	Y1.1	M42	Spindle gear 2 output signal	
CN62.11	Y1.2	M43	Spindle gear 3 output signal	
CN62.12	Y1.3	M44	Spindle gear 4 output signal	
CN62.13	Y1.4	M12(DOQPJ)	Outer chuck clamping output / Inner chuck unclamping output signal	
CN62.14	Y1.5	M13(DOQPS)	Outer chuck unclamping output /inner chuck clamping output signal	
CN62.15	Y1.6	TL+	Tool post forward rotation output signal	
CN62.16	Y1.7	TL-	Tool post reverse rotation output signal	
CN62.29	Y2.0		Tool post motor braking signal (Yantai AK31)/ tool post unclamping output (Liuxin Tool Post)	

DB Pin	PLC address	Function defined by standard PLC address		Remark
CN62.30	Y2.1		Tool post pre-indexing electromagnet signal (Yantai AK31)/ Tool post lock output (Liuxin Tool Post)	
CN62.31	Y2.2	YLAMP	Tri-colored lamp – yellow (normal state, non-running, non-alarm)	
CN62.32	Y2.3	GLAMP	Tri-colored lamp – green (running state)	
CN62.33	Y2.4	RLAMP	Tri-colored lamp – red (alarm state)	
CN62.34	Y2.5	M10	Tailstock advancing output signal	
CN62.35	Y2.6	M11	Tailstock retracting output signal	
CN62.36	Y2.7		Reserved	
CN62.37	Y3.0		Reserved	
CN62.38	Y3.1		Reserved	
CN62.39	Y3.2		Reserved	
CN62.40	Y3.3		Reserved	
CN62.41	Y3.4	SORI	Spindle orientation signal	
CN62.42	Y3.5	SEC0	Spindle orientation selection signal 1	
CN62.43	Y3.6	SEC1	Spindle orientation selection signal 2	
CN62.44	Y3.7	SEC2	Spindle orientation selection signal 3	
CN62.17～CN62.19 CN62.26～ CN6228			0V	
CN62.20～CN62.25			+24V	

Note1: Addresses X0.0~X0.7,X1.0~X1.7,X2.0~X2.7,X3.0~X3.7 are valid at a high-level, i.e. when the input signal +24V is connected, the state of address X signal is 1; when disconnected, the state is 0.

Note 2: When the state of address Y signal is 1, the output signal is connected to 0V (0V output); when the sate of address Y signal is 0, the output signal is at high-impedance state.

6.2.2 MPG Interface

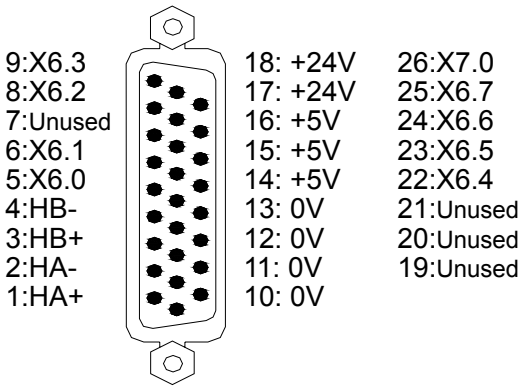
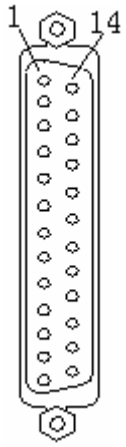


Fig. 6.2.3 CN31 MPG (26-pin, D type, Male)

DB Pin	Signal Definition	Signal Instruction	Function defined by standard PLC address
CN31.1,CN31.2	HA+，HA-	MPG phase A signal input	/
CN31.3,CN31.4	HB+，HB-	MPG phase B signal input	/
CN31.5	X6.0	PLC signal address, binary input	External hand-held unit X axis selection signal
CN31.6	X6.1	PLC signal address, binary input	External hand-held unit Y axis selection signal
CN31.8	X6.2	PLC signal address, binary input	External hand-held unit Z axis selection signal
CN31.9	X6.3	PLC signal address, binary input	External hand-held unit ×1 gear signal
CN31.22	X6.4	PLC signal address, binary input	External hand-held unit ×10 gear signal
CN31.23	X6.5	PLC signal address, binary input	External hand-held unit ×100 gear signal
CN31.24	X6.6	PLC signal address, binary input	External hand-held unit ×1000 gear signal
CN31.25	X6.7	PLC signal address, binary input	External hand-held unit the 4th axis selection signal
CN31.26	X7.0	PLC signal address, binary input	External hand-held unit the 5th axis selection signal
CN31.10，CN31.11 CN31.12，CN31.13	0V	0V	/
CN31.14，CN31.15 CN31.16	+5V	+5V	/
CN31.17,CN31.18	+24V	+24V	/

Note: X6.0~X7.0 input are valid at high-level, i.e. when the input signal is connected to +24V, the input is valid and the state of X address is 1; when disconnected, the state of X address is 0.

6.2.3 Spindle Interface



1: SCP+
2: SDIR+
3: GND
4: SALM
5: X5.0 (VPO)
6: X5.1 (SAR/PAR)
7: SRDY
8: X5.2 (ZSP)
9: GND
10: SPC
11: +24V
12: AGND
13: SVC

14: SCP-
15: SDIR-
16: GND
17: +24V
18: SSET
19: SEN
20: Y5.0 (VP)
21: Y5.1 (TAP)
22: Y5.2 (SFR)
23: Y5.3 (SRV)
24: GND
25: AGND

Fig. 6.2.4 CN15 the 5th axis ·spindle interface (25-pin, D type, female)

DB Pin	Signal Definition	Signal Instruction	Function defined by standard PLC address
CN15.1,CN15.14	SCP+，SCP-	Command pulse signal	/
CN15.2,CN15.15	SDIR+,SDIR-	Command direction signal	/
CN15.4	X5.3 (SALM)	Drive unit alarm signal	/
CN15.7	SRDY	Servo ready signal	/
CN15.18	SSET	Pulse disabled signal	/
CN15.19	SEN	Axis enable signal	/
CN15.10	SPC	Zero point signal	/
CN15.13	SVC	Spindle analog voltage output	/
CN15.12,CN15.25	AGND	Spindle analog voltage output common port	/
CN15.5	X5.0(VPO)	PLC signal address, binary input	Spindle speed/position state signal
CN15.6	X5.1(SAR/PAR)	PLC signal address, binary input	Spindle position/speed arrival signal
CN15.8	X5.2(ZSP)	PLC signal address, binary input	Spindle zero-speed output signal
CN15.20	Y5.0(VP)	PLC signal address, binary input	Spindle speed/position switch signal
CN15.21	Y5.1(TAP)	PLC signal address, binary input	Spindle speed loop gain 2 selection signal, used for tapping
CN15.22	Y5.2(SFR)	PLC signal address, binary input	Spindle CCW signal
CN15.23	Y5.3(SRV)	PLC signal address, binary input	Spindle CW signal
CN15.11,CN15.17	+24V	+24V	/
CN15.3,CN15.9, CN15.16,CN15.24	GND	0V Binary signal grounding	/

Note 1: X5.0, X5.1, X5.2 input are valid at a low level, i.e. when the input signal is connected to 0V, the input is valid and X address state is 1; when disconnected, the state is 0. Please note that the valid low-level input is different with the I/O addresses X0~X3 in general machine.
Note 2: When the state of Y address signal is 1, the output signal is connected to 0V (0V output); when the state is 0, the output signal is at high-impedance state.

6.2.4 Standard Operation Panel

(1) Address X

Address defined by PLC	Corresponding Key on the Panel	Remark
X18.0	Block skip	
X18.1	Auxiliary lock	
X18.2	Spindle override increase	
X18.3	Single block	
X18.4	Machine tool lock	
X18.5	Dry run	
X18.6	Spindle override decrease	
X18.7	Spindle override 100%	
X19.0	C axis moves along – direction(C -) /MPG C	
X19.1	C/S switch	
X19.2	Cycle start	
X19.3	Tailstock	
X19.4	The 4^{th} axis moves along – direction (4^{th} -)/MPG 4^{th}	
X19.5	Z axis moves along – direction (Z-)/ MPG Z	
X19.6	Y axis moves along – direction (Y-)/ MPG Y	
X19.7	X axis moves along – direction (X-)/ MPG X	
X20.0	Protection door	
X20.1	Tool post forward rotation	
X20.2	Tool offset	
X20.3	Tool post reverse rotation	
X20.4	Cooling	
X20.5	Spindle stop	
X20.6	Manual rapid traverse	
X20.7	Optional stop	
X21.0	Program restart	
X21.1	Spindle CW	

X21.2	Spindle jog	
X21.3	Spindle CCW	
X21.4	The 4th axis moves along + direction (4th+)	
X21.5	C axis moves along + direction (C+)	
X21.6	Spindle exact stop	
X21.7	Feed hold	
X22.0	MPG mode	
X22.1	Space key on the right of DNC	
X22.2	MANUAL mode	
X22.3	MDI mode	
X22.4	DNC mode	
X22.5	AUTO mode	
X22.6	REFERENCE POINT RETURN mode	
X22.7	EDIT mode	
X23.0	Rapid traverse override 100%/MPG×1000	
X23.1	Z axis moves along + direction (Z+)	
X23.2	Rapid traverse 50%/ MPG×100	
X23.3	Rapid traverse 25%/ MPG×10	
X23.4	Y axis moves along + direction (Y+)	
X23.5	Rapid traverse F0/ MPG×1	
X23.6	X axis moves along + direction (X+)	
X23.7	Hydraulic pressure	
X24.0	Space key below the cycle start	
X24.1	Chuck	
X24.2	Lubrication	
X24.3	Space key on the right of spindle CCW	
X24.4 ～ X24.7	Undefined	System reserved
X25.0 ～ X25.7	Connected to terminal strip	Reserved for user
X26.0 ～ X26.7	Connected to terminal strip	Reserved for user
X27.0 ～ X27.7	Connected to terminal strip	Reserved for user
X28.0	Connected to terminal strip	Connected to panel baud switch (spindle override OV 1)
X28.1	Connected to terminal strip	Connected to panel baud switch (spindle override OV 2)
X28.2	Connected to terminal strip	Connected to panel baud switch (spindle override OV 3)
X28.3	Connected to terminal strip	Connected to panel baud switch (spindle override OV 4)
X28.4	Connected to terminal strip	Connected to panel baud switch (feedrate override OV1)

X28.5	Connected to terminal strip	Connected to panel baud switch (feedrate override OV2)
X28.6	Connected to terminal strip	Connected to panel baud switch (feedrate override OV3)
X28.7	Connected to terminal strip	Connected to panel baud switch (feedrate override OV4)
X29.0	Connected to terminal strip	Connected to panel button (cycle start)
X29.1	Connected to terminal strip	Connected to panel button (feed hold)
X29.2	Connected to terminal strip	Connected to panel key switch button (program protection lock)
X29.3	Connected to terminal strip	Connected to panel knob normally-open terminal (spindle rotation allowed)
X29.4	Connected to terminal strip	Connected to panel knob normally-closed terminal (feed allowed)
X29.5 ～ X29.7	Connected to terminal strip	Reserved for user

Note: The PLC address X18~X24 are the fixed addresses input by keys on the panel, and their functions are fixed. Addresses X25~X29 are lead to the terminal strip on the backboard of the panel, the exact functions are defined by the PLC run in the system.

(2) Address Y

Address defined by PLC	Corresponding key on the panel	Remark
Y18.0	Block skip indicator	
Y18.1	Auxiliary lock key indicator	
Y18.2	L5 indicator	
Y18.3	Single block indicator	
Y18.4	Machine lock key indicator	
Y18.5	Dry run key indicator	
Y18.6	C/S switch key indicator	
Y18.7	C/S axis – direction key indicator	
Y19.0	C axis + direction (C+) key indictor	
Y19.1	The 4^{th} axis + direction (4^{th}+) key indicator	
Y19.2	Cycle start key indicator	
Y19.3	Feed hold key indicator	
Y19.4	Program restart key indicator	
Y19.5	Optional stop key indicator	

Y19.6	Spindle override decrease key indicator	
Y19.7	Spindle override 100% key indicator	
Y20.0	Spindle override increase indicator	
Y20.1	Hydraulic pressure key indicator	
Y20.2	Tailstock key indicator	
Y20.3	Lubrication key indicator	
Y20.4	Protection door key indicator	
Y20.5	Tool post forward rotation key indicator	
Y20.6	Tool offset key indicator	
Y20.7	Tool post reverse rotation key indicator	
Y21.0	Digitron (right) output (value 1)	
Y21.1	Digitron (right) output (value 2)	
Y21.2	Digitron (right) output (value 4)	
Y21.3	Digitron (right) output (value 8)	
Y21.4	Digitron (left) output (value 1)	
Y21.5	Digitron (left) output (value 2)	
Y21.6	Digitron (left) output (value 4)	
Y21.7	Digitron (left) output (value 8)	
Y22.0	MPG mode indicator	
Y22.1	Indicator of space key on the right of DNC	
Y22.2	MANUAL mode indicator	
Y22.3	MDI mode indicator	
Y22.4	DNC mode indicator	
Y22.5	AUTO mode indicator	
Y22.6	REF. mode indicator	
Y22.7	EDIT mode indicator	
Y23.0	Rapid traverse override 100% indicator	
Y23.1	Z axis + direction indicator	
Y23.2	Rapid traverse override 50% indicator	
Y23.3	Rapid traverse override 25% indicator	
Y23.4	Y axis + direction indicator	
Y23.5	Rapid traverse override F0 indicator	
Y23.6	X axis + direction indicator	
Y23.7	System alarm (ALM) indicator	
Y24.0	Cooling key indicator	
Y24.1	Chuck key indicator	
Y24.2	Indicator of space key on the right of the spindle CCW key	
Y24.3	Spindle exact stop key indicator	
Y24.4	Spindle stop key indicator	

Y24.5	Spindle CW key indicator	
Y24.6	Spindle jog key indicator	
Y24.7	Spindle CCW key indicator	
Y25.0	The 4th – direction key indicator	
Y25.1	Z axis – direction (Z-) key indicator	
Y25.2	Y axis – direction (Y-) key indicator	
Y25.3	Z axis machine zero point indicator	
Y25.4	Y axis machine zero point indicator	
Y25.5	Z axis machine zero point indicator	
Y25.6	X axis – direction (X-) key indicator	
Y25.7	Rapid traverse key indicator	
Y26.0	Indicator of space key below the cycle start key	
Y26.1	L4 indicator	
Y26.2	L3 indicator	
Y26.3	L2 indicator	
Y26.4	L1 indicator	
Y26.5	System running (RUN) indicator	
Y26.6	C axis machine zero point indicator	
Y26.7	4th axis machine zero point indicator	
Y27.0～Y27.7	Connected to terminal strip	Reserved for user
Y28.0～Y28.7	Connected to terminal strip	Reserved for user
Y29.0	Connected to terminal strip	Connected to panel button indicator (cycle start)
Y29.1	Connected to terminal strip	Connected to panel button indicator (feed hold)
Y29.2～Y29.7	Connected to terminal strip	Reserved for user

Note: The PLC addresses Y18~Y26 are the fixed addresses of indicator output on the panel; their functions fixed. Addresses Y27~Y29 are lead to the terminal strip on the backboard of the panel; the exact functions are defined by PLC.

6.3 Standard PLC Functions

6.3.1 Cycle Start and Feed Hold

The standard operation panel consists of a group of keys and a group of external buttons which is used to realize the function of cycle start and feed hold. Please note the difference between addresses of keys and buttons.

- **Address definition**

X0019							BIT2		
X0021		BIT7							

X19.2: Input address of cycle start key on the panel

X21.7: Input address of feed hold key on the panel

Y0019						BIT3	BIT2		

Y19.2: Output address of cycle start indicator on the panel

Y19.3: Output address of feed hold indicator on the panel

X0029								BIT1	BIT0

X29.0: Input address of external cycle start button

X29.1: Input address of external feed hold button

Y0029								BIT1	BIT0

Y29.0: Output address of external feed hold button indicator

Y29.1: Output address of external cycle start button indicator

Control logic

When the system is in automatic running state, press feed hold key or external feed hold button, the running process will be suspended.

When the system is in stop or suspended state, press cycle start key or external cycle start button, the automatic running will be performed.

6.3.2 Feed/Spindle Hold

- **Address definition**

X0029					BIT4	BIT3			

X29.3: Input address of feed enabled (connected to the feed/spindle knob)

X29.4: Input address of spindle knob enabled (connected to the feed/spindle knob)

- **Control parameter**

K0010						KNEN			

K10.3 =1: The function of feed hold knob on the machine tool is enabled;

=0: The function of feed hold knob on the machine tool is disabled;

Control logic

The feed/spindle hold knob can enable the spindle rotation and cycle start;

When the spindle is rotating, and the knob is set to the spindle hold position, the spindle output is disabled.

When the spindle is not rotating, and the knob is set to the spindle hold position, the spindle cannot be started.

When the knob is set to feed hold position during automatic running, the feed stops and “Dwell” is displayed.

When the knob is set to feed hold position during automatic running, press “Cycle Start” button, the program execution is disabled.

6.3.3 Program Lock

- **Address definition**

X0029						BIT2		

X29.2: Input address of program protection signal

Control parameter

K0009								RPRT

K9.0 =1: Program lock is shielded

=0: Program lock is not shielded

- **Control logic**

When K9.0 is set to 1, the program lock is disabled, regardless of the signal X39.2; and both the program and parameter writing are enabled.

When K9.0 is set to 0, the program lock is enabled.

When signal X29.2 is valid, both program and parameter writing are enabled.

When signal X29.2 is invalid, both program and parameter writing are disabled.

6.3.4 Feedrate Override

- **Address definition**

X0028	BIT7	BIT6	BIT5	BIT4				

X28.4: Feedrate override signal OV0

X28.5: Feedrate override signal OV1

X28.6: Feedrate override signal OV2

X28.7: Feedrate override signal OV3

- **Control logic**

It adopts digital code rotary switch; the code is two's complement.

6.3.5 Spindle Override

- **Address definition**

X0018	BIT7	BIT6				BIT2		

X18.2: + Spindle override +

X18.6: Spindle override -

X18.7: 100% Spindle override 100%

Y0019	BIT7	BIT6						
Y0020								BIT0

Y19.6: Spindle override – key indicator

Y19.7: Spindle override 100% key indicator

Y20.0: Spindle override + key indicator

Relevant parameter

DT0023	Spindle override indicator flicker period (100-1000ms)

- **Control logic**

①When the spindle override is greater than 100%:

Override <120%: spindle override+indicator flickers; the flicker period is set by DT23

Override = 120%: spindle override + indicator normally lights up.

② When the spindle override equals to 100%

Spindle override 100% key indicator normally lights up;

③ when spindle override is less than 100%:

Override > 50%: spindle override – key indicator flickers, the flicker period is set by DT23;

Override= 50%: spindle override – key indicator normally lights up;

Note: When thread cutting is performing, spindle override is disabled.

6.3.6 Spindle CCW/CW Control

- **Address definition**

Y0000	SPZD		M5	M4	M3			

Y0.3: Spindle CCW output signal (M3)

Y0.4: Spindle CW output signal (M4)

Y0.5: Spindle stop signal (M5)

Y0.7: Spindle braking output signal (SPZD)

X0020			BIT5					
X0021					BIT3		BIT1	

X20.5: Spindle stop key

X21.1: Spindle CW key

X21.3: Spindle CCW key

Y0024	BIT7		BIT5	BIT4				

Y24.4: Spindle stop indicator

Y24.5: Spindle CW indicator

Y24.7: Spindle CCW indicator

- **Control parameter**

K0010							BIT1	

K10.1 =1: When the system is reset, the output signals M03, M04, M08, M32 are NOT OFF

K10.1 =0: When the system is reset, the output signals M03, M04, M08, M32 are OFF.

DT0005	MTIME
DT0010	SPDDLT
DT0011	SPZDTIME

DT05: the execution time of M code (ms); value range: 100~5000ms

DT10: M05 and the delay time (ms) of spindle braking output; value range:0~10000ms

DT11: Spindle braking output time; value range: 50~60000ms

➢ **Motion sequence**

The sequence of spindle motion is shown as follows:

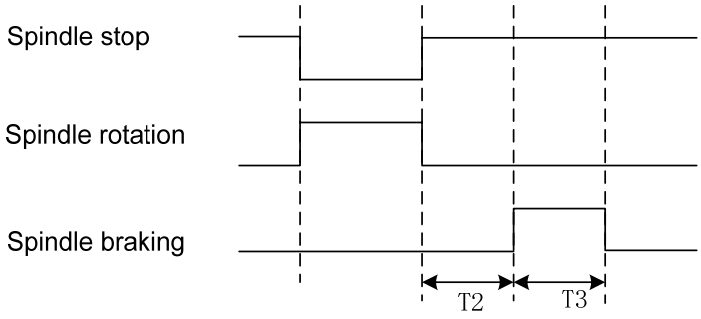


Fig. 6-3-1 Sequence diagram of spindle CCW/CW

Note: T2 is the delay time counting from the issuing of spindle stop signal to spindle braking signal; T3 is the spindle braking duration.

➢ **Logic control**

After power-on, M05 output is enabled.

When M05 is enabled, and M03 or M04 is executed, M03 or M04 output is enabled and remains unchanged; meanwhile, M05 output is disabled.

When M03 or M04 is enabled, and M05 is executed, M03 or M04 output is disabled and M05 output is enabled and remains unchanged.

When M03 or M04 output is enabled, the execution of M04 or M03 will lead to system alarm.

The delay time of spindle braking signal SPZD is set by parameter DT0010, and the duration is set by DT0011.

Note: when CNC performs emergency stop, signal M03 or M04 output is disabled, and signal M05 is output at the same time.

6.3.7 Spindle Jog

➢ **Address definition**

X0024		BIT6						

X24.6: Spindle jog mode signal

➢ **Control parameter**

DT12	spindle jog duration (ms)

➢ **Logic control**

In increment, MPG or MANUAL mode, press 

to enable the spindle jog mode; press key 

the spindle rotates CCW; press 

, spindle rotates CW; the rotating duration is set by PLC parameter DT12.

6.3.8 Spindle 8-Point Pre-Orientation

- **Address definition**

Y0003		SEC2	SEC1	SEC0	SORI				

Y3.4: Spindle orientation signal

Y3.5～Y3.7: Spindle orientation selection signal

X0002		COIN							

X2.7: Spindle orientation completed signal

- **Control logic**

① After commands M51~M58 are executed, PLC issues orientation selection signals SEC0, SEC1, SEC2 to Drive to determine the position.

② After 40ms delay, PLC issues spindle orientation signal SORI to Drive;

③ Drive starts orientation;

④ After the orientation is finished. Drive sent the spindle orientation completed signal COIN to PLC;

⑤ If the PLC does not receive the COIN signal in 6000ms after the orientation selection signal is issued, the system will issue an alarm “spindle orientation time is too long”.

⑥ The spindle can be in rotating or stop state before the orientation, and it will be in stop state after the orientation.

- **Control sequence diagram**

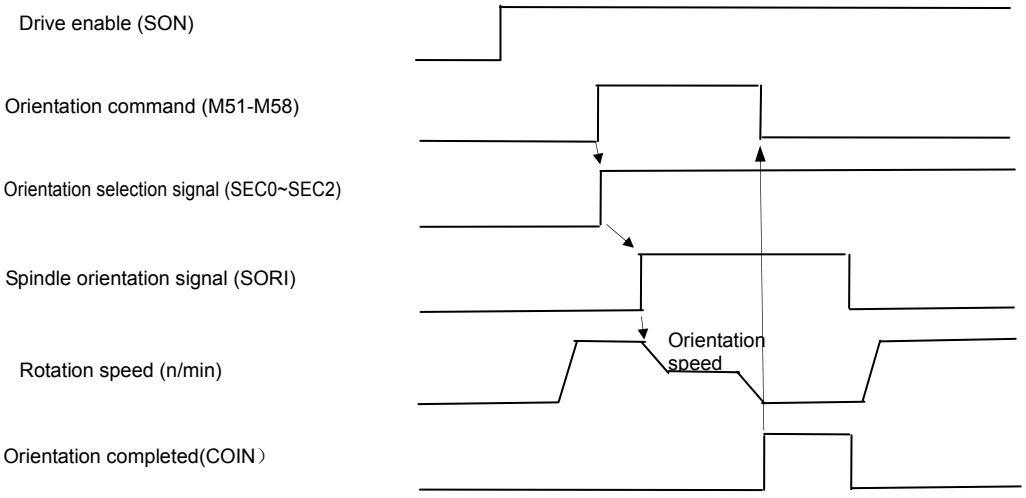


Fig. 6-3-2 Pre-orientation sequence

- **8-Point Pre-Orientation Method:**

① Connect the system and servo drive unit correctly. The interfaces includes: enable signal interface (SON), forward/reverse rotation interface (SRF/SRV), position start signal interface (STAO), position completed signal interface (COIN) and multi-point orientation selection input signal (SP0~SP2). Refer to the spindle servo manual for details.

② First, set the parameters related to servo spindle drive unit correctly.

a) Set the servo spindle drive unit parameter No. PA4 (i.e. speed control mode) to 1.

b) PA55 sets the spindle speed during orientation.

c) PA56 sets the position screen during orientation.

d) When PA66 selects the orientation, the signal of selected encode is used as position feedback input signal;

e) PA67 sets the spindle encoder line numbers according to the actual machine tool configuration.

③ Bring up the monitoring menu dP-APo, and press ↵, then, E □□□□ is displayed; symbol "E" means the spindle is at an uncertain position, its value cannot be the reference value for orientation.

④ The spindle should rotate one circle at least. After the drive unit detects the Z pulse signal of the spindle encoder, and finds the right position, dP-APo will turn into ⊦ □□□□, which indicates that the position of current encoder is right.

⑤ Adjust the spindle to orientation point gradually, then record the displayed position in dP-APo; write it in parameter PA58, then save the parameter. This reference value is position 1.

⑥ User can adjust 8 orientation points continuously and the position of the points will be recorded and written in parameters PA58~65 by sequence. In this method, multi-point orientation is performed.

The relationship between speed selection input signals(SP0, SP1, SP2) and parameters PA58~PA56 is shown in the table below:

Output signals / Commands	SEC2 (Y3.7)	SEC1 (Y3.6)	SEC0 (Y3.5)	Corresponding positions
M51	0	0	0	Orientated position 1 (PA58)
M52	0	0	1	Orientated position 2 (PA59)
M53	0	1	0	Orientated position 3 (PA60)
M54	0	1	1	Orientated position 4 (PA61)
M55	1	0	0	Orientated position 5 (PA62)
M56	1	0	1	Orientated position 6 (PA63)
M57	1	1	0	Orientated position 7 (PA64)
M58	1	1	1	Orientated position 8 (PA65)

⑦ Enable the drive unit (input SON signal and SFR signal), input the orientation start signal (STAO), and keep the low-level signal valid, the servo motor runs at the speed set by PA55; after the orientation point is found (determined by SP0~SP2), the servo motor keeps at the position and outputs orientation completed signal (COIN).

⑧ After the system detected COIN signal, the orientation completion is confirmed. Cancel the orientation start signal to proceed to the next operation.

6.3.9 Spindle Speed Binary Control

➢ **Address definition**

Y0001					S04/M44	S03/M43	S02/M42	S01/M41

S01～S04: Spindle speed binary control signal

M41～M44: Spindle automatic gear changing signal

➢ **Control parameter**

K0010				BIT4				

K10.4 =1: Gear spindle—the spindle speed is controlled by gears

K10.4=0: Analog spindle—the spindle speed is controlled by analog voltage

➢ **Control logic**

When K104 is set to 1, the spindle speed binary control is enabled.

When CNC is turned on, S1~S4 outputs are disabled.

When one of the commands S01, S02, S03, S04 is executed, the corresponding S signal is valid and remains the same, meanwhile, the output of the rest three signals is cancelled, i.e. among S1~S4, only one is valid at a time.

When S00 command is executed, the output S1~S4 is cancelled.

6.3.10 Spindle Gear Control

➢ **Address definition**

Y0001					S04/M44	S03/M43	S02/M42	S01/M41

M41～M44: Automatic spindle gear change output signal

X0001		M42I	M41I					

M41I～M42I: The in-position signal when spindle changes to 1 or 2 gear automatically.

Control parameter

K0010				BIT4				
K0013	AGER	AGIN	AGIM	ASTR				

K10.4 =1: Gear spindle—the spindle speed is controlled by gear stage.

K10.4 =0: Analog spindle—the spindle speed is controlled by analog voltage control.

K13.4 =1: Automatic spindle gear change is valid.

K13.4 =0: Automatic spindle gear change is invalid.

K13.5 =1: Checks the gear change in-position signal during automatic spindle gear change

K13.5 =0: Does not check the gear change in-position signal during automatic spindle gear change.

K13.6 =1: Gear change in-position signal is valid when connected to +24V

K13.6 =0: Gear in-position signal is valid when disconnected to +24V.

K13.7 =1: Spindle gear stage is stored when power-off.

K13.7 =0: Spindle gear stage is not stored when power-off.

3741	The maximum spindle speed at gear stage 1 (MSG1)
3742	The maximum spindle speed at gear stage 2 (MSG2)
3743	The maximum spindle speed at gear stage 3 (MSG3)
3744	The maximum spindle speed at gear stage 4 (MSG4)

MSG1, MSG2, MSG3, MSG4: When spindle analog voltage output is 10V, they correspond to the maximum spindle speeds at gear stages 1, 2, 3, 4. When spindle automatic gear change is valid, they corresponds to the spindle speed commanded by M41, M42, M43, M44; when the spindle gear stage is not stored after power-off, the default setting is gear stage 1 after CNC is power-on.

DT0000	Automatic gear change signal output delay time 1 (SFT1TME)
DT0001	Automatic gear change signal output delay time 2 (SFT2TME)

➢ **Function description**

The spindle automatic gear change function is used to control the spindle mechanical gear automatic switch; when CNC executes S____ command, it calculates the analog voltage which is output to spindle servo or inverter, according to the parameters (No. 3741~No. 3744) of current gear stages controlled by M4n.

6.3.11 Cooling Control

➢ **Address definition**

Y0000								M08
X0020				BIT4				
Y0024								BIT0

Y0.0: Cooling signal output (M08)

X20.4: Cooling key input

Y24.0: Cooling key indicator

➢ **Function description**

After CNC is power-on, M09 is valid, which means M08 output is disabled.

When M08 is executed, M08 output is enabled and the cooling pump is ON; when M09 is executed, M08 output is cancelled and the cooling pump is OFF.

Press the cooling key 

on the operation panel, the M08 output state is inverted.

Note 1: When emergency stop or M30 is executed, M08 output is cancelled and cooling is OFF.
Note 2: When CNC is reset, the bit 1 of K10 sets whether the M08 output is cancelled or not.
Note 3: M09 corresponds to no output signal. When M09 is executed , M08 output is cancelled and the cooling is OFF.

6.3.12 Lubricating Control

➢ **Address definition**

Y0000							M32	
X0024						BIT2		
Y0020					BIT3			

Y0.1: Lubrication output signal (M32)

X24.2: Lubrication key

Y20.3: Lubrication key indicator

DT0013	Manual lubrication output time
DT0016	Automatic lubrication interval time
DT0017	Automatic lubrication output time

DT13: Lubrication duration (0~60000ms); when it is set to 0, lubrication output state is unchanged.

DT16: Automatic lubrication interval time (0~60000ms)

DT17: Automatic lubrication output time (0~60000ms)

➢ **Function description**

The lubrication function defined by GSK988T standard PLC program includes two kinds: non-auto-lubrication and auto-lubrication. When DT16=0 or DT17=0, the auto-lubrication function is disabled.

a) Non-auto-lubrication

When DT>0, lubrication output is executed at regular time. The key 

on the panel is enabled or when M32 is executed, lubrication Y0.1 output is valid, meanwhile, the indicator signal Y20.3 output is valid. When the time set by DT13 ends, lubrication Y0.1 and Y20.3 output is cancelled; if M33 is executed before the time approaches, the lubrication Y0.1 output and Y20.3 output is cancelled.

When DT13=0, the lubrication output is inverted. The key 

is enabled or when M32 is executed, lubrication Y0.1 output is valid, meanwhile, indicator signal Y20.3 output is valid; When key 

is enabled again or M33 is executed, lubrication Y0.1 output is OFF, meanwhile, indicator signal Y20.3 is OFF.

b) Auto-lubrication

When DT16 > 0, DT17>0, the system starts to countdown for the duration set by DT16 after system power-on, then, the lubrication output is performed. When the time set by DT17 ends, the lubricating stops, and so forth. During automatic lubrication, 

key and M32, M33 commands are valid in the interval time, and they are disabled in the lubrication output time.

Note 1: During emergency stop or the execution of M30, M32 output will be cancelled and the lubrication is OFF.

Note 2: When CNC is reset, the bit 1 of K10 sets whether the M32 output is cancelled or not.

Note 3: M33 corresponds to no output signal. When M33 is executed , M32 output is cancelled and the cooling is OFF.

6.3.13 Chuck Control

- **Address definition**

Y0001			DOQPS	DOQPJ				

Y1.4: Outer chuck clamping/inner chuck unclamping output

Y1.5: Outer chuck unclamping/inner chuck clamping output signal

X0000						DIQP		
X0003				NQPS	WQPJ			

X0.2 : Chuck control input signal (DIQP)

X3.3: Outer chuck clamping in-position/inner chuck unclamping in-position signal (WQPJ)

X3.4: Outer chuck unclamping in-position/inner chuck clamping in-position signal (NQPJ)

X0024							BIT1	
Y0024							BIT1	

X24.1: Chuck key

Y24.1: Chuck key indicator

- **Control parameter**

K0013							SLSP	SLQP

K13.0 = 1: Chuck control function is enabled.

K13.0=0: Chuck control function is disabled.

K13.1=1: When chuck function is enabled, the system checks whether the chuck is clamping.

K13.1=0: When the chuck function is enabled, the system does not check whether the chuck is clamping or not; If the chuck is unclamping, the spindle cannot be started.

K0014						PB2		PB1

K14.0 = 1: Check chuck in-position signal

K14.0 =0: Does not check chuck in-position signal

K14.2 = 0:Outer chuck mode, WQPJ is outer chuck clamping signal, NQPJ is outer chuck unclamping signal

K14.2 =1: Inner chuck mode, NQPJ is inner chuck clamping signal, WQPJ is inner chuck unclamping signal.

- **Control logic**

Signals in outer chuck mode	Clamping	WQPJ(X3.3): Chuck clamping in-position signal
		DOQPJ (Y1.4): Chuck clamping output signal
	Unclamping	NQPJ(X3.4): Chuck unclamping in-position signal
		DOQPS (Y1.5): Chuck unclamping output signal
Signals in inner chuck mode	Clamping	NQPJ(X3.4): Chuck clamping in-position signal
		DOQPS (Y1.5): Chuck clamping output signal
	Unclamping	WQPJ(X3.3): Chuck unclamping in-position signal
		DOQPJ (Y1.4): Chuck unclamping output signal

When then system is power-on, the signals DOQPJ and DOQPS is the state before power-off last time. i.e. DOQPJ and DOQPS are stored when power-off.

When chuck control input (DIQP) is valid or the key 

is pressed, the chuck clamping/unclamping signal is output alternatively, i.e. each time the chuck control input signal is enabled, the output state changes.

When the spindle is rotating, DIQP input and chuck key on the panel are disabled; M13 cannot be executed, and an alarm will occur. The output state will not change.

In reset or emergency stop state, the output state of DOQPJ, DOQPS remain unchanged.

- **Sequence diagram:**

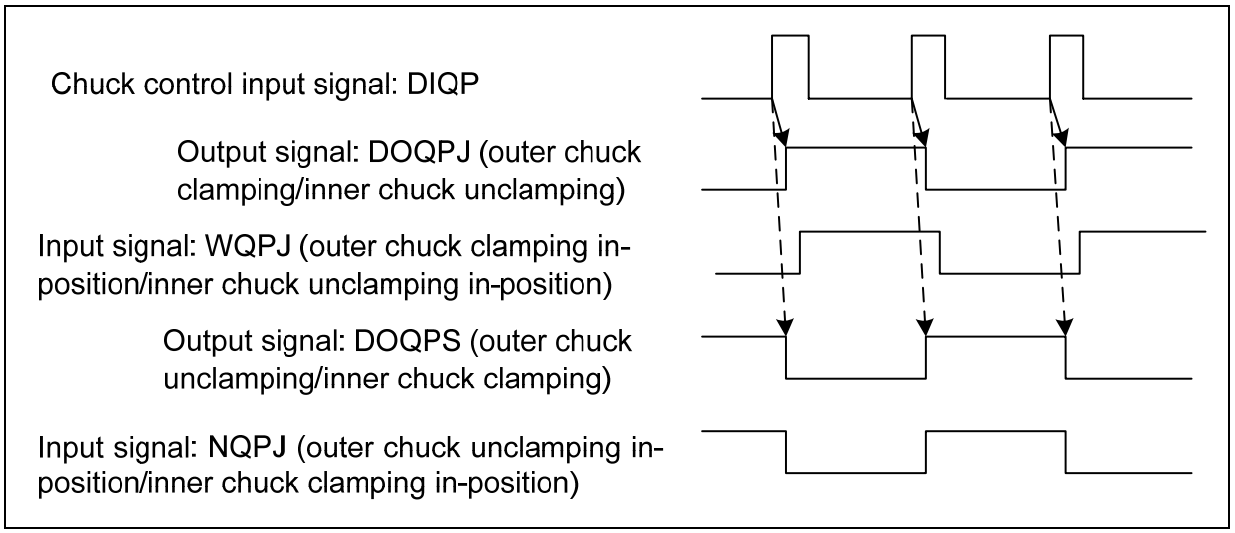


Fig.6-3-2 Chuck control sequence diagram

The control logic for signals K13.1 and K14.0:

When K13.1=1, K14.0=0:

After chuck clamping signal is output, the spindle can be started, otherwise, system alarm will be issued.

When K13.1＝1， K14.0＝1:

When the chuck clamping signal and in-position signal are valid, the spindle can be started, otherwise, system alarm will be issued.

When K13.1＝0，K14.0＝0:

No matter the chuck is clamping or not, spindle can be started.

When K13.1＝0，K14.0＝1:

When the chuck in-position signal is valid, the spindle can be started, otherwise, system alarm will be issued.

6.3.14 Tailstock Control

- **Address definition**

Y0002			M11	M10				

Y2.5: Tailstock advancing output signal (DOTWJ)

Y2.6: Tailstock retracting output signal (DOTWS)

X0000					DITW			

X0.4: Tailstock control input signal

X0019					BIT3			

X19.3: Tailstock key on the panel

Y0020						BIT2		

Y20.2: Tailstock key indicator on the panel

➢ **Control parameter**

K0013						SLTW		

K13.2 =1: Tailstock control function enabled

K13.2 =0: Tailstock control function disabled

➢ **Sequence diagram**

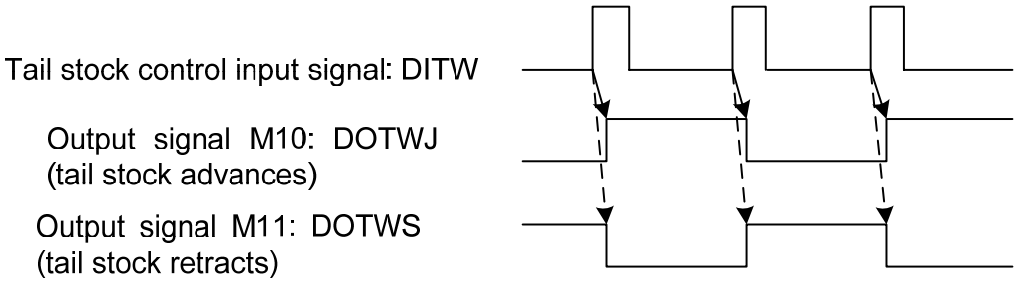


Fig. 6-3-3 Tail stock control sequence diagram

After power-on, the signals DOTWJ and DOTWS remain the state when power-off the previous time, i.e. DOTWJ and DOTWS are stored when power-off.

When tailstock control input (DITW) or the key 

are enabled, the tailstock advance/retract signal is output alternatively, i.e. each time when the tailstock control input signal is valid, the output state changes.

After M10 is executed, the signal DOTWJ is output, and the tailstock advances; after M11 is executed, signal DOTWS is output, and the tailstock retracts.

When the spindle is rotating, the tailstock control DITW input and tailstock key on the panel are disabled; the execution of M11 is invalid and will trigger an alarm. The output state keeps the same.

When CNC is reset or in emergency stop state, the output state of signal DOTWJ/DOTWS remains unchanged.

6.3.15 Low Pressure Detection

➢ **Address definition**

X0000		PRES						

X0.6: Low pressure detection signal (PRES)

➢ **Control parameter**

K0014			BIT5	BIT4				

K14.4 =0: High-level alarm; When PRES is connected to 24V, the low pressure alarm is issued.
K14.4 =1: Low-level alarm; When PRES is disconnected with 24V, the low pressure alarm is issued.
K14.5 =0: Low pressure detection function is disabled.
K14.5 =1: Low pressure detection function is enabled.

DT0002	Low pressure alarm detection time

The delay time before the low pressure alarm is issued: 0~60000ms

- **Function description**

When the low pressure detection function is selected, the PRES signal is valid. As the delay time set by DT0002 passed, CNC issues an alarm; meanwhile, the feed axis stops, spindle stops and the automatic cycle function cannot be started. Press RESET key or turn off the power to cancel the alarm.

6.3.16 Overtravel Signal of Axes

- **Address definition**

X0003						LMI3+	LMI2+	LMI1+
X0004		LMI5-	LMI5+	LMI4-	LMI4+	LMI3-	LMI2-	LMI1-

X3.0: 1st axis + direction overtravel signal X4.0: 1st axis – direction overtravel signal
X3.1: 2nd axis + direction overtravel signal X4.1: 2nd axis – direction overtravel signal
X3.2: 3rd axis + direction overtravel signal X4.2: 3rd axis – direction overtravel signal
X4.3: 4th axis + direction overtravel signal X4.4: 4th axis – direction overtravel signal
X4.5: 5th axis + direction overtravel signal X4.6: 5th axis – direction overtravel signal

- **Control parameter**

K0010						BIT2		

K10.2 =1: Low-level signal of each axis is valid
K10.2=0: High-level signal of each axis is valid

6.3.17 Tool Change Control

The tool post control logic supported by standard ladder diagram is realized through the combination of Bit 7, Bit 6, and Bit2 of parameter K.

K0011	BIT7	BIT6				BIT2		

K11.6=0, K11.7=0: Standard tool change method (select tool change method A or B by K11.2)
K11.2=1: Tool change method A (adopts Jingcheng Tool Post)
K11.2=0: Tool change method B (adopts Changzhou Tool Post)
K11.6=1, K11.7=0: Adopts Yantai Tool Post AK31 Series (8-position, 10-position, 12-position tool post)
K11.6=0，K11.7=1: Adopts Taiwan Liuxin 8-position hydraulic tool post

➢ **Control parameter**

K0011	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

K11.0= 1: The tool post lock signal is low-level;
K11.0 = 0: The tool post lock signal is high-level;
K11.1= 1: The tool position signal is low-level;
K11.1 = 0: The tool position signal is high-level;
K11.3= 1: Check the tool position signal after tool change
K11.3=0: Does not check the tool position signal after tool change
K11.4 = 1: Check tool post lock signal
K11.4 = 0: Does not check tool post lock signal

DT0007	Delay time from the tool post forward rotation to reverse rotation output (0-1000ms)
DT0008	Alarm time when the TCP signal is not received (0-1000ms)
DT0009	Tool post reverse rotation lock time (0-1000ms)

Note: K11.0,K11.1 ,K11.3, K11.4,DT0007,DT0008,DT0009 are used for the control in standard tool change method only (when the standard tool change mode is selected i.e. K11.6=0, K11.7=0, these parameters are valid).

➢ **Address definition**

K11.7	K11.6	K11.2	Tool Post Type	Address to be Used
0	0	1	Tool change method A	X1.7(T1),X2.0(T2),X2.1(T3),X2.2(T4),X0.7(T5),X1.0(T6), X1.1(T7),X1.2(T8),X2.6(TCP),Y1.6(TL+),Y1.7(TL-)
0	0	0	Tool change method B	
0	1	/	Yantai Tool Post AK31 Series (8, 10, 12-position)	X1.7(T1), X2.0(T2), X2.1(T3), X2.2(T4), X2.6 (lock proximity switch signal), X0.7(tool post pre-indexing proximity switch), X1.0 (tool table strobe signal), X1.1(tool table overheat detection), Y1.6(TL+), Y1.7(TL-), Y2.0(TZD tool table braking),Y2.1(tool table pre-indexing electromagnet)
1	0	/	Liuxin Hydraulic Tool Post LS120 (8-position)	X1.7(T1), X2.0(T2), X2.1(T3), X2.2(T4), X0.7(tool post stop and lock sensor),X1.0(tool post releasing\lock output sensor),Y2.0(tool post releasing output) ,Y2.1(tool post lock output),Y1.6(tool post forward rotation output),Y1.7(tool post reverse rotation output)

➢ **Control logic**

a) K11.7=0，K11.6=0，K11.2=1: Tool change method A

- In MANUAL, MDI or AUTO mode, tool change is executed, CNC outputs the tool post forward rotation signal (TL+) and detects the tool position signal. After the tool position signal is detected, CNC turns OFF the tool post forward rotation signal (TL+), and check whether the tool position signal transition occurs, if it does, the tool post reverse rotation signal (TL-) is output and then turned OFF after the time set by PLC parameter

DT009 ends.

- If the Bit 4 of K0011 is set to 1 (lock detection signal), the system detects the tool post lock signal. If the TCP signal is not received within the time set by PLC parameter DT008, a system alarm will be generated.
- If the Bit 3 of K0011 is set to 1 (tool position check signal after the tool change), when the tool post reverse rotation time ends, confirm the consistency of the current tool position input signal and current tool number; if they are not consistent, the system will issue an alarm.
- The tool change is finished.

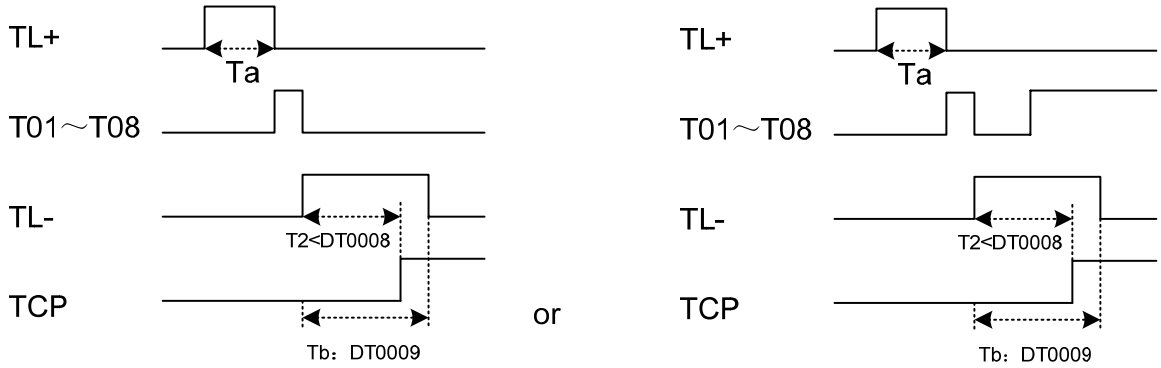


Fig. 6-3-4 Tool change A sequence diagram

b) K11.7=0，K11.6=0，K11.2=0： B Tool change method B

When the tool change is executed, the system outputs the tool post forward rotation signal TL+, and detects the tool position signal. After the tool position signal is detected; TL+ output is turned OFF. When the time set by PLC parameter DT007 ends, the tool post reverse rotation signal TL- is output; when the time set by parameter DT009 ends, the tool post reverse signal TL- is turned OFF.

- If the Bit 4 of K0011 is set to 1 (lock detection signal), the system starts to detect the tool post lock signal; if the system does not receive TCP signal within the time set by parameter DT008, an alarm will be generated.
- If the Bit3 of K0011 is set to 1 (tool position check signal after tool change), when the tool post reverse rotation time ends, confirm the consistency of the current tool position input signal and current tool number; if they are not consistent, the system will issue an alarm.
- The tool change is finished.

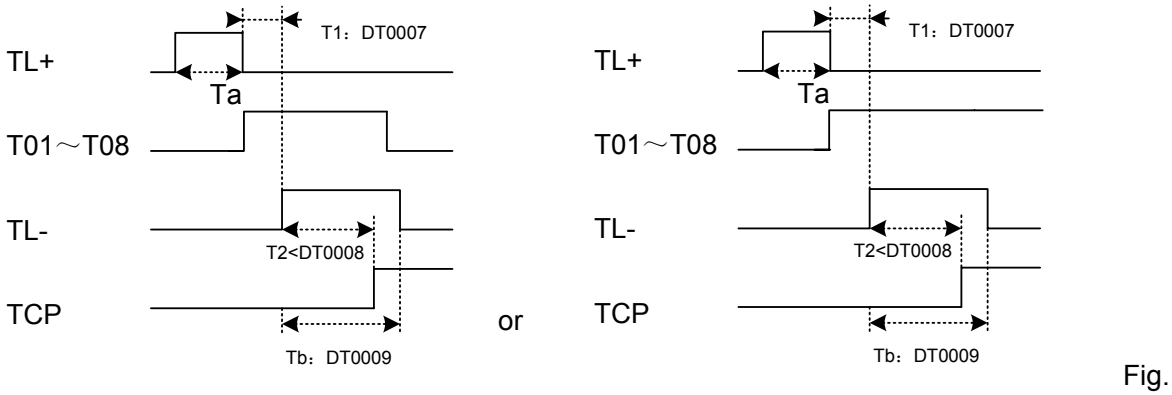


Fig. 6-3-5 Tool change B sequence diagram

c) K11.7=0，K11.6=1: Yantai Tool Post AK31 Series

1) Tool change process

- Confirm that the tool post braking signal TZD is OFF.
- The system determines the shortest path according to object tool number and current tool number, and selects the output rotation direction under the principle of “select the nearest tool”, and determines the output signal is TL+ or TL-. Then, as the tool post rotates, the tool selection begins.
- In the process of rotation, the system decodes according to the tool position encode signal T1~T4, and identifies the current tool number. When the tool post rotates to the position before the object one, the system starts to detect the transition of tool post strobe signal. The strobe signal transition of the tool position before the object position is from on to off. The system outputs the tool post pre-indexing electromagnet signal, the tool post pre-indexing electromagnet supplies power.
- When the detected tool post pre-indexing proximity switch input signal is at high level, turn off the tool post rotation output signal (TL+ or TL-), and the motor stops running.
- After delaying 50ms, the system outputs a signal (TL- or TL+) which is inverted to the original rotation direction, then, the tool post rotates in a reversed direction.
- When the detected tool post lock proximity switch input signal is at high level, turn off the tool post rotation output signal (TL+ or TL-), the motor stops running, then, the system outputs tool post braking signal (TZD), the motor braking device is energized.
- After delaying 200ms, turn off the tool post pre-indexing electromagnet output signal, the tool post pre-indexing electromagnet is de-energized.
- When the current tool number is detected again, confirm the consistency of the current tool position encoder signal and object tool number.
- Confirm that the lock proximity switch signal is at high level again.
- If the steps listed above are correct, turn off the tool braking signal TZD, the tool change is finished.
- In the process of tool change, if the motor overheat signal is detected, an alarm is raised, and all signals output are turned OFF.

2) Tool change flow chart

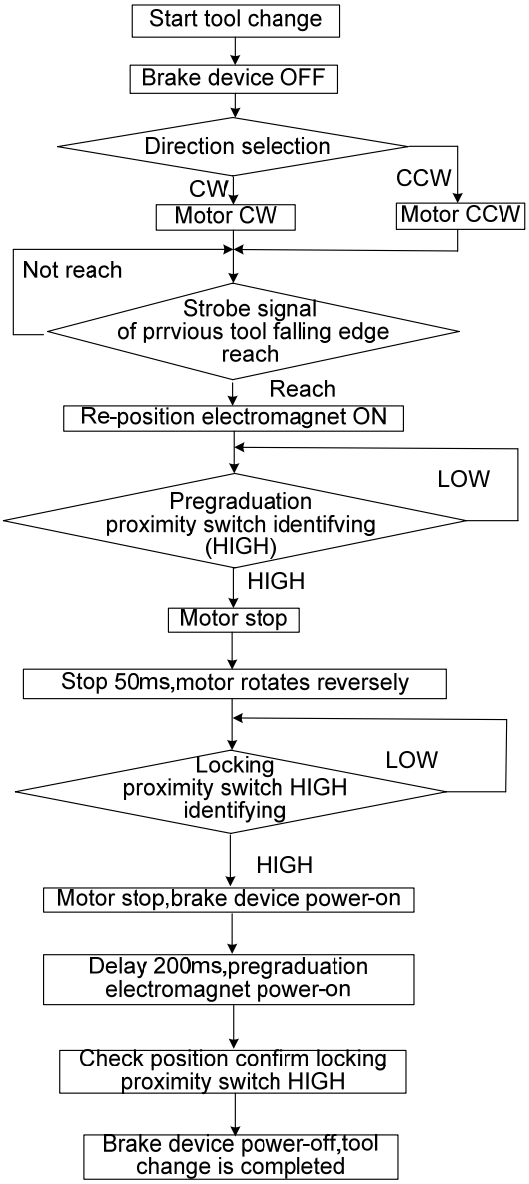


Fig. 6-3-6 AK31 Tool change flow chart

c) K11.7=1，K11.6=0: Liuxin 8-position hydraulic tool post

1) Input/output configuration

Sensor A: Tool position detection sensor→T1(X1.7): tool position signal

Sensor B: Tool position detection sensor→T2(X2.0): tool position signal

Sensor C: Tool position detection sensor→T3(X2.1): tool position signal

Sensor D: Tool position detection senso→T4(X2.2): tool position signal

Sensor E: Tool post rotation stop and lock sensor→SSE(X0.7): tool post rotation stop and lock signal

Sensor F: tool post release/lock signal output sensor→SSF(X1.0): tool post release/lock signal

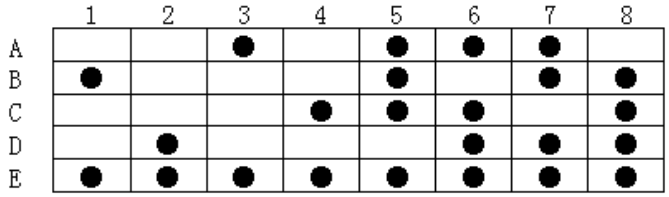
Sol A: tool post release/lock magnetic valve→ Y2.0: tool post release output

Y2.1: tool post lock output

Sol B: tool post forward/reverse rotation magnetic valve→TL+(Y1.6):tool post forward rotation output

→TL-(Y1.7): tool post reverse rotation output

2) Position and signal table



3) Signal instruction

Sensor A,B,C,D: provide tool position detection; but no motion signal is issued.

Sensor E: each time a tool is changed, the tool post stop and lock signal is issued. When the tool post rotates to the desired position, Sensor E induces and cut off the power of rotation magnetic valve, making the tool post rotation stopped, then, it starts the tool post lock magnetic valve to ensure that the tool post is locked.

Sensor F: Release/lock confirmation signal; When Sensor F does not induce, i.e. the tool post is released, tool post rotation can be started; when Sensor F induces, i.e. the tool post is locked, the tool change is finished.

Sol A: Controls the tool post release/lock

Sol B: Controls the tool post forward/reverse rotation

4) Description of tool change process

Example: Tool is changed from No.1 to No. 4

Step 1: Sol A is energized (tool post released)

Step 2: Confirm that the Sensor F does not induce, Sol B is energized, oil hydraulic motor rotates.

Step 3: Start to detects the tool position signal (Note: Sensor E induces at tool position 1, 2, 3, but when the position 4 is not reached, the motion of lock is not performed; when the tool position 3 signal is confirmed, set the anticipation of Sensor E, when the tool post rotates to the position 4, Sensor E induces i.e. cut off the power of Sol B, tool post stops rotation; meanwhile Sensor E controls the Sol A to lock the tool post.

6.3.18 Emergency Stop

- **Address definition**

X0000			ESP					

X0.5: Emergency stop input signal

- **Control parameter**

K0010	ESP							

K10.7 =1: External emergency stop input signal (X0.5) high-level alarm

K10.7 =0: External emergency stop input signal (X0.5) low-level alarm

3009	ESP							

3009#7 =1: Emergency stop alarm is raised when external emergency stop signal (X0.5) is 1.

3009#7 =0: Emergency stop alarm is raised when external emergency stop signal (X0.5) is 0.

Note: The values of k10.7 and No. 3003#7 should be set consistently.

6.3.19 Tri-Colored Lamp

- **Address definition**

Y0002					BIT4	BIT3	BIT2	

Y2.2: Tri-colored lamp –yellow, normal state (non-running, non-alarm state)

Y2.3: Tri-colored lamp – green, running state

Y2.4: Tri-colored lamp – red, alarm state

- **Control parameter**

K0012			LAMP					

K12.5 =1: Tri-colored lamp output function is valid

K12.5=0: Tri-colored lamp output function is invalid.

6.4 Standard PLC Parameter Instruction

6.4.1 Parameter K

Note: K0~K7 do not need to be set.

Address	Parameter meaning	Initial value
K8.0	X axis manual movement direction (1: reversed, 0: not reversed)	0
K8.1	Y axis manual movement direction (1: reversed, 0: not reversed)	0
K8.2	Z axis manual movement direction (1: reversed, 0: not reversed)	0
K8.3	The 4th axis manual movement direction (1: reversed, 0: not reversed)	0
K8.4	C axis manual movement direction (1: reversed, 0: not reversed)	0
K9.0	Shield program protection lock (1: shield, 0: does not shield)	0
K9.7	Alarm occurs when invalid M code is commanded (1: yes, 0: no)	0
K10.0	Feed override (1: inverted, 0: not inverted)	0
K10.1	Turn off the spindle, cooling and lubrication output during reset (1: No, 0: Yes)	0
K10.2	Axes overtravel input signal alarm level (1:low-level alarm, 0: high-level alarm)	0
K10.3	Machine panel feed/spindle enable knob (1:valid, 0: invalid)	1
K10.4	Spindle type (1: gear, 0: analog)	0
K10.7	External emergency stop input signal (X0.5) (1: high-level alarm, 0: low-level alarm)	0
K11.0	Tool post lock signal (1: low-level, 0: high-level)	0
K11.1	Tool position signal (1: low-level, 0: high-level)	0
K11.2	Tool change method when standard tool change mode is selected (1: method A, 0: method B)	1

K11.3	Check tool position signal after tool change (1: Yes, 0: No)	0
K11.4	Check tool post lock signal (1: Yes, 0: No)	1
K11.6	Tool post selection (PB8 PB7: 00 standard tool post/01 Yantai Tool Post/10 Liuxin Tool post)	0
K11.7	Tool post selection (PB8 PB7: 00 standard tool post/01 Yantai Tool Post/10 Liuxin Tool post)	0
K12.0	1/0: manual inverted tool change is valid/invalid	0
K12.2	Zero return direction locked automatically (1: Yes, 0: No)	0
K12.5	Tri-colored lamp output function (1: enabled, 0: disabled)	0
K12.6	External hand-held unit (1: enabled, 0: disabled)	0
K12.7	Machine tool operation panel (1: MPU02B, 0: MPU02A)	0
K13.0	Chuck control function (1:enabled, 0:disabled)	1
K13.1	If the chuck function is valid, check the chuck clamping state when the spindle is started (1: Yes, 0: No)	1
K13.2	Tailstock control function (1: valid, 0: invalid)	0
K13.4	Spindle gear stage is stored when power-off (1: Yes, 0: No)	1
K13.5	Spindle automatic gear change in-position signal active level (1: low-level, 0: high-level)	0
K13.6	Check spindle automatic gear change in-position signal (1: Yes, 0: No)	0
K13.7	Spindle automatic gear change function (1: valid, 0: invalid)	0
K14.0	Check chuck clamping/unclamping signal (1:Yes, 0: No)	0
K14.2	Chuck mode (1: inner chuck, 0: outer chuck)	0
K14.4	Low-pressure alarm signal level (1: low-level alarm, 0: high-level alarm)	0
K14.5	Low-pressure alarm function (1: valid, 0: invalid)	0
K14.6	Protection door input signal alarm level (1: low-level alarm, 0: high-level alarm)	0
K14.7	Protection door alarm function (1: valid, 0: invalid)	0
K15.0	Starting up operation mode MD1	0
K15.1	Starting up operation mode MD2	0
K15.2	Starting up operation mode MD4	0
K15.4	Starting up operation mode (1: MD2, MD2, MD4, 0: the mode when power-off the last time)	0
K15.6	Servo spindle 8-point orientation function (1: valid, 0: invalid)	0

6.4.2 Parameter DT

DT address	PLC initial value	Minimum input value	Maximum input value	Meaning
DT0000	1000	0	60000	Spindle gear change time 1 (ms)
DT0001	1000	0	60000	Spindle gear change time 2 (ms)

DT0002	3000	0	60000	Low-pressure alarm detection time (ms)
DT0003	5000	100	5000	Tool change (for one tool position) time upper limit (ms)
DT0004	15000	1000	60000	Tool change (for maximum tool positions) time upper limit (ms)
DT0005	500	100	5000	M code execution duration (ms)
DT0006	500	100	5000	S code execution duration (ms)
DT0007	500	0	4000	Delay time of the tool post from forward rotation stop to reverse rotation output (ms)
DT0008	500	0	4000	Alarm time when the TCP signal is not received (ms)
DT0009	1000	0	4000	Tool post reverse rotation lock time (ms)
DT0010	0	0	10000	Delay time of M05 and spindle braking output (ms)
DT0011	50	0	60000	Spindle braking output time (ms)
DT0012	100	0	60000	Spindle jog time (ms)
DT0013	0	0	60000	Lubricating start time (0-60000ms) (0: no limit)
DT0016	0	0	60000	Automatic lubricating interval time (ms)
DT0017	0	0	60000	Automatic lubricating output time (ms)
DT0019	1000	100	60000	Chuck function execution duration when in-position signal is not checked (ms)
DT0021	1000	100	60000	Spindle stop, chuck operation enable delay time (ms)
DT0022	500	100	1000	Alarm indicator flickering period (100-1000) (ms)
DT0023	500	100	1000	Spindle override indicator flickering period (100-1000) (ms)
DT0024	400	100	2000	Feed override knob debounce time (ms)
DT0025	400	100	2000	Spindle override knob debounce time (ms); valid when the machine tool panel is MPU02B
DT0032	10000	0	60000	Liuxin 8-Position Hydraulic Tool Change alarm time (ms)
DT0034	10000	0	60000	AD31 Series Tool Post allowable continuous time upper limit (ms)
DT0035	1000	0	4000	AK31 Series Tool Post lock proximity switch signal detection time upper limit (ms)

6.4.3 Parameter DC

DC address	PLC initial value	Minimum input value	Maximum input	Meaning

			value	
DC0000	50	0	200	The output voltage value of inverter during spindle jog (0.01V)
DC0001	5	0	50	The output voltage value of inverter during spindle automatic gear change (0.01V)

6.4.4 Parameter D

D address	PLC initial value	Minimum input value	Maximum input value	Meaning
D0	4	1	16	Number of tools on a tool post
D1	1	0	5	Internal controlled axis number corresponding to X axis manual movement key (the key is invalid when it is set to 0)
D2	0	0	5	Internal controlled axis number corresponding to Y axis manual movement key (the key is invalid when it is set to 0)
D3	2	0	5	Internal controlled axis number corresponding to Z axis manual movement key (the key is invalid when it is set to 0)
D4	0	0	5	Internal controlled axis number corresponding to the 4th axis manual movement key (the key is invalid when it is set to 0)
D5	0	0	5	Internal controlled axis number corresponding to C axis manual movement key (the key is invalid when it is set to 0)

6.5 Signals G, F Used in Standard PLC

6.5.1 Signal G

Address	Function	Symbol
G4.3	Auxiliary function end signal	FIN
G4.4	The 2M function end signal	MFIN2
G4.5	The 3M function end signal	MFIN3
G5.0	Miscellaneous function end signal	MFIN
G5.2	Spindle function end signal	SFIN

Address	Function	Symbol
G5.3	Tool function end signal	TFIN
G5.6	Auxiliary function lock signal	AFL
G6.2	Manual absolute value signal	ABSM
G6.4	Override cancel signal	OVC
G7.2	Cycle start signal	ST
G7.4	Stroke check 3 release signal	RLSOT3
G7.6	Stored stroke limit selection signal	EXLM
G8.4	Emergency stop signal	ESP
G8.5	Feed dwell signal	SP
G8.7	External reset signal	ERS
G10,G11	Manual feedrate override signal	JV0～JV15
G12	Feedrate override signal	FV0～FV7
G14.0, G14.1	Rapid traverse override signal	ROV1,ROV2
G18.0～G18.3	MPG 1 feed axis selection signal	HS1A～HS1D
G18.4～G18.7	MPG 2 feed axis selection signal	HS2A～HS2D
G19.4，G19.5	MPG/STEP override signal	MP1,MP2
G19.7	Manual rapid traverse selection signal	RT
G27.0	The 1st spindle selection signal	SSW1
G27.1	The 2nd spindle selection signal	SSW2
G27.3	The 1st spindle stop signal	SSTP1
G27.4	The 2nd spindle stop signal	SSTP2
G27.7	Spindle contouring control switch signal	CON
G28.1，G28.2	The 1st spindle gear selection signal	GR11，GR12
G29.0，G29.1	The 2nd spindle gear selection signal	GR21，GR22
G28.7	The 2nd position encoder selection signal	PC2SLC
G29.4	Spindle speed arrival signal	SAR
G29.6	Spindle stop signal	SSTP
G30	Spindle override signal	SOV0～SOV7
G32.0～G32.7 G33.0～G33.3	Signal of the 1st spindle motor speed command input by PLC	R01I～R12I
G34.0～G34.7 G35.0～G35.3	Signal of the 2nd spindle motor speed command input by PLC	R01I2～R12I2
G33.7	The 1st spindle motor speed selection command signal	SIND
G35.7	The 2nd spindle motor speed selection command signal	SIND2
G43.0 ～ G43.2,G43.5,G43.7	Mode selection signal	MD1,MD2,MD4,DNC1,ZRN
G44.0	Optional block skip signal	BDT1

Address	Function	Symbol
G44.1	Machine lock for all axes signal	MIK
G46.1	Signal block signal	SBK
G46.7	Dry run signal	DRN
G100.0～G100.4	Feed axis and direction selection signal	+J1～+J5
G102.0～G102.4		-J1～-J5
G114.0～G114.4	Overtravel signal	+L1～+L5
G116.0～G116.4		-L1～-L5
G200.0	Spindle jog function signal	SPHD
G201	Current tool position signal	NT00～NT07

6.5.2 Signal F

Address	Function	Symbol
F0.4	Feed dwell signal	SPL
F0.5	Cycle start signal	STL
F0.6	Servo ready signal	SA
F0.7	Automatic running signal	OP
F1.0	Alarm signal	AL
F1.1	Reset signal	RST
F1.3	Assignment end signal	DEN
F1.4	The 1st spindle enable signal	ENB
F1.7	CNC ready signal	MA
F2.0	Inch input signal	INCH
F2.1	Rapid traverse signal	RPDO
F2.2	Constant surface speed cutting signal	CSS
F2.3	Thread cutting signal	THRD
F2.7	Dry run detection signal	MDRN
F3.0	STEP mode detection signal	MINC
F3.1	MPG mode detection signal	MH
F3.2	MANUAL mode detection signal	MJ
F3.3	MDI mode detection signal	MDI
F3.4	DNC mode detection signal	MRMT
F3.5	AUTO mode detection signal	MMEM
F3.6	EDIT mode detection signal	MEDT
F4.0	Optional block skip detection signal	MBDT1
F4.1	Machine lock for all axes detection signal	MMLK
F4.2	Manual absolute detection signal	MABSM
F4.3	Single block detection signal	MSBK
F4.4	Auxiliary function lock detection signal	MAFL
F4.5	Machine zero return mode detection	MREF

Address	Function	Symbol
	signal	
F7.0	Auxiliary function strobe signal	MF
F7.2	Spindle speed function strobe signal	SF
F7.3	Tool function strobe signal	TF
F8.4	The 2M auxiliary function strobe signal	MF2
F8.5	The 3M auxiliary function strobe signal	MF3
F9.4	M decoding signal	DM30
F9.5		DM02
F9.6		DM01
F9.7		DM00
F10～F13	Auxiliary function code signal	M00～M99
F14～F15	The 2M auxiliary function code signal	M200～M299
F16～F17	The 3M auxiliary function code signal	M300～M399
F22～F25	Spindle speed code signal	S00～S31
F26～F29	Tool function code signal	T00～T31
F36.0～F26.7 F37.0～F37.3	The 1st spindle S12-digit code signal	R01O～R12O
F38.2	The 2nd spindle enable signal	ENB2
F40～F41	The 1st spindle actual speed signal	AR00～AR15
F62.7	Object parts counting reach signal	PRTSF
F94.0～ F94.4	Machine zero return end signal	ZP1～ZP5
F96.0～ F96.4	The 2nd reference point machine zero return end signal	ZP21～ZP25
F98.0～ F98.4	The 3rd reference point machine zero return end signal	ZP31～ZP35
F100.0～ F100.4	The 4th reference point machine zero return end signal	ZP41～ZP45
F102.0～ F102.4	Axis movement signal	MV1～MV5
F106.0～ F106.4	Axis moving direction signal	MVD1～MVD5
F120.0～ F120.4	Reference point setting signal	ZRF1～ZRF5
F200.0～F200.7 F201.0～F201.3	The 2nd spindle S12-digit code signal	R01O2～R12O2
F202～F203	The 2nd spindle actual speed signal	AR002～AR152

APPENDIX A ALARM LIST

A.1 Program Alarms (P/S Alarms)

No.	Message	Contents
000	Emergency stop, ESP open circuit	Restore the ESP emergency stop signal input to cancel the alarm.
001	Part program open failure	Press RESET key to cancel the alarm, or turn on the power again.
002	More than 256 character in a program line	Too many characters in a program line; modify the program.
003	Data exceeds the permitted range	The input data exceeds the permitted range or 8 digits. Modify the program.
004	Address not found	No address but only digits or characters are input at the head of a block. Modify the program.
005	No data followed the address	Data is not followed the address or the expression format is erroneous (bracket is not used). Modify the program.
006	Incorrect usage of minus sign	The minus sign "-" is used incorrectly (used in improper place or overused). Modify the program.
007	Incorrect usage of decimal point	The decimal point "." is used incorrectly (used in improper place or overused). Modify the program.
008	Illegal address input	Unusable address is input in significant area. Modify the program.
009	Incorrect G code	An unusable G code or G code corresponding to a not provided function is specified. Modify the program.
010	Address duplication error	The same addressed appears more than once in a block. Alternatively, a block contains two or more G codes belonging to the same group. Please refer to parameter 3403#6 AD2. Modify the program.
011	Command cannot run in DNC occurs	Command that cannot run in DNC is found. Modify the program.
012	Too many M codes	Specifying multiple M codes in the same block is not allowed. Please refer to parameter 3404#7 M3B. Modify the program.
014	Divided by zero	Division by zero is specified (including tan 90°). Modify the program.
017	Parameter writing failure	Please check whether the parameter file is in normal state. Pay attention that the user area may be corrupted.
018	Part program operation failure	Press “RESET” key to cancel the alarm.
019	End of record	The end of record (%) is specified, or the program end is not specified. Please refer to parameter 3404#6 EOR. Modify the program.

No.	Message	Contents
020	DNC time out	DNC transmission failure; Please check it.
021	The setting value of feedrate is not within the range.	During cutting feed, the feedrate is not specified or incorrectly specified. The values for modal G98 and G99 are different. Please check modal G98 and G99, and modify the program.
022	The setting value of spindle speed is out of the range.	Spindle rotational speed or surface speed is set incorrectly. Please refer to parameter 3031 SCB and modify the program.
023	M command value is out of the range.	A wrong M code is specified. Please refer to parameter 3030 MCB and modify the program.
024	G code usage error	The G code needs to be used independently, and cannot coexist with other G codes in the same block. Modify the program.
025	Illegal tool number	An inexistent tool number is specified. Please refer to parameter 3032 TCB and modify the program.
026	Illegal offset number	The offset number of the selected tool position offset value by T code is too large. Modify the program.
027	Illegal offset valid	The offset value selected by T code is too large. Modify the program.
028	T code is not allowed in this block.	G50, G10 and G04 cannot be specified in the same block with T codes. Please refer to parameter 5006#1 TGC and modify the program.
031	Too many axes commanded	The number of commanded axes exceeds that of simultaneously controlled axes. Modify the program.
032	Axis that cannot perform interpolation command is specified.	Axis not included in the selected plane is specified or the basic axis and the parallel axis are specified simultaneously. Modify the program.
033	Illegal plane axis commanded	In circular interpolation, axis not included in the selected plane is specified. Modify the program.
034	No arc radius command	In circular interpolation, neither R nor I, J, K is specified. Refer to parameter 3403#5 CIR and modify the program.
035	Illegal radius command	In circular interpolation, address R value is wrong. Please refer to parameter 3403#4 RER and modify the program.
036	Exceeds the radius difference range	In circular interpolation, the difference between the distance from the start point to the center point and distance from the end point to the center point exceeds the value set by parameter. Please refer to parameter and modify the program.
037	Thread run-out length J, K value commanded incorrectly in thread cutting.	The run-out length exceeds the permitted range. K value is less than zero in G32, G34 commands; J value or K value is less than zero in G92 command. Modify the program.
038	Illegal lead command	Lead command value F is out of the range; or in

No.	Message	Contents
		variable lead thread cutting, the lead variation exceeds the range. Modify the program.
039	In thread cutting command, the thread run-out length of long axis is excessive.	The thread run-out length of long axis exceeds the thread cutting length. Modify the program.
040	In thread cutting command, the thread run-out length of short axis is excessive.	In G92 command, the thread run-out length of the short axis exceeds the distance from the starting point to the end point.
041	Illegal plane selection	In plane selection command, more than one parallel axes are specified in the same direction. Modify the program.
042	Metric/inch conversion command error	The metric/inch conversion command is not specified alone in a line, or is not specified in the first line. Metric/inch conversion is performed when a subprogram is called. Modify the program.
043	Reference point return uncompleted	The reference point return cannot be performed normally because the reference point return start point is too close to the reference point or the speed is too slow. Separate the start point far enough from the reference point, or specify a sufficient fast speed for reference point return.
044	Reference point return uncompleted	When automatic operation dwells, manual reference point return cannot be performed.
045	Axes not on reference point	During the reference point return check (G27), the specified axis does not return to the reference point. Modify the program.
046	G28 found in sequence return	A program restart command is specified without the execution of reference point return after power-on or emergency stop, and G28 is found during research. Perform the reference point return.
047	The specified axis does not return to the reference point.	The specified axis does not return to the reference point by cycle start. Please perform reference point return.
048	Wrong reference point commanded	In G30 block, a value other than 2~4 is specified by P. Modify the program.
051	G37 reach signal not acquainted	In the automatic tool compensation function (G36, G37), the measurement position reach signal (XAE or EAE) is not turned on in the area specified by parameter. This may be caused by a setting or operation error.
052	Offset number not found in G37	G36, G37 automatic tool compensation is specified without T code. Modify the program.
053	T coded not allowed in G37	T code and automatic tool compensation (G36, G37) are specified in the same block. Modify the program.
054	Illegal axis command in G37	In automatic tool compensation function (G36, G37),

No.	Message	Contents
		invalid axis is specified, or command is incremental, or the γ value in automatic tool compensation is less than ε. Modify the program.
055	G37 command invalid	The automatic tool compensation function (G36, G37) is invalid. Check parameter No. 6240#7 IGA and modify the program.
058	G31 cannot be used in feed-per-rotation mode	In feed-per-rotation mode, skip cutting command is specified. Modify the program.
059	G31 cannot be used in tool nose radius compensation mode	Skip cutting command is specified in tool nose radius compensation mode. Modify the program.
061	Illegal P specified in G10	The P value which specifies the offset value is too large, or no P value is specified. Modify the program.
062	Illegal compensation value in G10	The specified offset value is too large. Modify the program.
063	Illegal format in G10 or L50	Any of the following errors occurs in the specified format at the programmable-parameter input 1) address N or R is not entered 2) a number not specified for a parameter is entered. 3) The axis number is too large. 4) An axis number is not specified in the axis-type parameter. 5) An axis number is specified in the parameter which is not an axis type.
065	The offset value are accumulated excessively	In G50 offset accumulation, the accumulated offset exceeds the permitted range. Modify the program.
068	Stroke check range setting error	In the stroke check range set by command G22, the specified positive coordinate value or parameter value is not greater than the negative coordinate value, or the differential between them is less than 2000 least input increment. Check parameter No. 1322 or No. 1323, and modify the program.
071	Spindle orientation unexecuted	The spindle indexing is performed before spindle orientation. Execute the spindle orientation.
072	C/H code and movement command in the same block	Spindle indexing command C, H and other axis movement command are in the same block. Modify the program.
073	M code and movement command in the same block	Spindle indexing M code and other axis movement command are in the same block. Modify the program.
074	Illegal command G12.1/G13.1	When the polar coordinate interpolation is started or cancelled, the condition is wrong. 1). G12.1/G13.1 is specified in the mode that is not specified by G40. 2). Error is found in plane selection. The parameter

No.	Message	Contents
		specifying is erroneous. Modify the program or the parameter.
075	An unusable G code is specified in polar coordinate interpolation.	An unusable G code is specified in polar coordinate interpolation. Modify the program.
081	Undefined address P	In the programs commanded by M98, G65, G66, the address P (program number) is not defined. Modify the program.
082	Subprogram nesting error	The nested subprogram exceeds 12 levels.
083	Program number not found	In the blocks that contain M98, M99, G65, G66, the program number specified by address P is not found. Modify the program.
084	Subprogram call error	A higher-level program or the subprogram itself is called by M98, G66 or G66. Modify the program.
085	Program call statement cannot be run in MDI and DNC modes.	Macro program call and subprogram call in MDI and DNC modes are not supported. Modify the program.
090	Axis command error in spindle constant surface speed control mode	In modal G96, the basic calculation axis commanded by parameter does not exist. Modify the program.
101	Spindle speed is too fast during thread cutting	During thread cutting, the spindle speed is so fast that the feeding axis cannot run normally. Modify the program.
121	Canned cycle command is specified in non-ZX plane.	The canned cycle command is not specified in the basic ZX coordinate system. Modify the program.
122	Axis not included in the basic ZX coordinate system is specified in canned cycle.	Axis not included in the ZX coordinate system is specified in canned cycle. Modify the program.
123	The R value (radius value) is greater than the U value (absolute value) in G90, G92 commands.	In G90, G92 commands, when the plus or minus signs for the R and U are different, the absolute value of R value (radius value) is greater the U value (absolute value). Modify the program.
124	In G94 command, the R absolute value is greater than W absolute value	In G94 command, when the plus or minus signs for the R and W are different, the absolute value of R is greater that of the W value. Modify the program.
126	Illegal plane selection in multiple-cycle command	Cycle command is not specified in ZX plane. Modify the program.
127	Axis not included in the ZX plane is specified in G70~G76.	Axis not included in the ZX plane is specified in G70~G76 commands or the G70~G76 loop. Modify the program.
128	Incorrect G code in G70~G73	An unusable G code is commanded between the two blocks which is specified by addresses P and Q in G70~G73. Modify the program.
129	G70~G73 commands cannot be run in MDI mode	G70~G73 commands including addresses P, Q are specified in MDI mode.

No.	Message	Contents
130	Macro statement execution is not allowed in G70~G73 loop	Macro statement execution is not allowed in G70~G73 loop. Modify the program.
131	Subprogram is called in G70~G73 loop	Subprogram cannot be called in G70~G73 loop. Modify the program.
132	Subprogram is called in G70~G73 command lines	Subprogram cannot be called in G70~G73 command lines. Modify the program.
133	In G70~G73 commands, the addresses P or Q is out of the range	In G70~G73 commands, the addresses P or Q is undefined or out of the range. Modify the program.
134	The sequence number not found in G70~G73 commands	The sequence number specified by address P or Q is not found in G70~G73 commands. Modify the program.
135	P and Q commands error in G70~G73 commands	In G70~G73 commands, the command values of P and Q are the same. Modify the program.
136	Two continuous blocks is not found in G71~G73 commands	Two continuous blocks is not found in G71~G73 commands, which will cause error. Modify the program.
137	In G71~G73 commands, the number of Ns-Nf blocks exceeds 100.	In G71~G73 commands, the Ns-Nf blocks are excessive. Modify the program.
138	In G71~G73 commands, the Ns-Nf blocks are non-monotonic	In multiple cycle command (G71 or G72), a non-monotonic object structure is defined; or in G73 cycle, the Z axis is non-monotonic; when the Z axis is set with retraction amount or finishing allowance, the X axis is non-monotonic. Please refer to parameter 5102#1 MRC and modify the program.
139	The orientation point commanded by G71~G73 is within the cutting range	When the orientation point commanded by G71~G73 is within the cutting range, tool collision may occur. Please refer to parameter 5104#2 FCK and modify the program.
141	In G73 cycle, the tool retraction direction of X axis is inconsistent with the finishing allowance direction.	In G73 cycle, the tool retraction direction of X axis is opposite to the finishing allowance direction. Modify the program.
142	In G73 cycle, the tool retraction direction of Z axis is inconsistent with the finishing allowance direction.	In G73 cycle, the tools retraction direction of Z axis is opposite to the finishing allowance direction. Modify the program.
143	Finishing allowance in G70~G73 exceeds the range	Finishing allowance of G70~G73 exceeds the range. Modify the program.

No.	Message	Contents
144	G00 or G01 is not commanded in starting block of the G71~G72 loop.	G00 or G01 needs to be commanded in starting block of the G71~G72 loop. Modify the program.
145	None of G00-G03 is commanded in starting block of the G73 loop	G00, G01, G02 or G03 is not commanded in the starting block of the G73 loop. Modify the program.
146	Only X axis increment is needed in the starting block of G71 loop	X axis is not commanded in the starting block of the G71 loop, or the X axis increment is zero, or Z axis is commanded. Modify the program.
147	Only Z axis increment is needed in the starting block of G72 loop	Z axis is not commanded in the starting block of the G71 loop, or the Z axis increment is zero, or X axis is commanded. Modify the program.
148	The single feeding amount in G71 or G72 command is less than zero	The single feeding amount in G71 or G72 command is less than zero. Modify the program.
149	The single tool retraction amount R(e) in G71 or G72 command is less than zero	The single retraction amount R(e) in G71 or G72 command is less than zero. Modify the program.
150	The total cutting amount in G73 exceeds the permitted range	The total cutting amount in G73 exceeds the permitted range. Modify the program.
151	The number of repetition R(d) in G73 command is out of the permitted range	The number of repetition R(d) in G73 command is less than 1 or greater than 000 after rounding. Modify the program.
152	Z axis command is not input in G74	Z axis command is not input in G74. Modify the program.
153	Q value in G74 is not in the range	Q value in G74 is not in the needed range. Modify the program.
154	X axis command is not input in G75	X axis command is not input in G75. Modify the program.
155	P value in G75 is not in the range	The P value in G75 is not in the needed range. Modify the program.
156	R(e) is less than zero in G74 or G75 command	Single tool retraction amount R(e) is less than zero in G74 or G75 command. Modify the program.
157	R(△d) is less than zero in G74 or G75 command	The tool retraction amount R(△d) is less than zero in G74 or G75 command when the cutting feed reaches the end point. Modify the program.
158	In G74 or G75, single cutting amount exceeds the range	In G74 or G75, the single cutting amount along Z or Z direction exceeds the permitted range. Modify the program.
160	In G76, X or Z axis movement amount is 0	In G76, X or Z axis movement amount is 0. Modify the program
161	The repetition number of G76 is less than 1 or greater than 99	The repetition number of G76 is less than 1 or greater than 99. Modify the program.

No.	Message	Contents
162	In G76, the thread chamfering angle exceeds the permitted range	In G76, the thread chamfering angle exceeds the permitted range. Modify the program.
163	In G76, the tool nose angle exceeds the permitted range	In G76, the tool nose angle exceeds the permitted range. Modify the program.
164	In G76, Q(△dmin) exceeds the permitted range	In G76, the minimum cut-in amount Q(△dmin) exceeds the permitted range. Modify the program.
165	The G76 finishing allowance R(d) exceeds the permitted range	The G76 finishing allowance R(d) is less than a minimum increment. Modify the program.
166	During taper thread cutting commanded by G76, the R value and U value are unmatched	During taper thread cutting commanded by G76, the start point of machining is between the thread start point and thread end point. Modify the program.
167	Thread height P value is not specified in G76 command	Thread height P value is not specified in G76 command. Modify the program.
168	The G76 thread height is less than the finishing allowance or the minimum cutting amount	The G76 thread height is less than the finishing allowance or the minimum cutting amount. Modify the program.
169	The Q value in G76 command is not within the range	The first cutting depth is not defined in G76 command: the Q value is not within the range or not input. Modify the program.
180	Illegal S command in rigid tapping	The S code in rigid tapping is undefined or out of the range. Modify the program.
181	Illegal K command in rigid tapping	The specified repetition number K value is out of the range in rigid tapping. Modify the program.
182	Illegal F command in rigid tapping	The cutting feedrate value is F is out of the range in rigid tapping. Please check the modal G98 and G99 and modify the program.
183	Program error in rigid tapping	The M code and S value is not in the same block in rigid tapping. Modify the program.
184	Illegal axis operation in rigid tapping	A move axis is specified between the M code and G84 command in rigid tapping. Modify the program.
185	The spindle cannot perform rigid tapping	In rigid tapping, the spindle is not selected. Modify the parameter.
186	Plane alteration during rigid tapping	During rigid tapping, a non-G18 plane is switched or the rigid tapping is enabled in non-G18 plane. Modify the program.
187	Data error in rigid tapping	The specified distance is too short or too long in rigid tapping. Modify the program.
188	Data repetition in rigid tapping	The same M code or S code is repeated between M code and G84 in rigid tapping. Modify the program.
189	M code repetition in rigid tapping	In rigid tapping, the M code cannot be in the same block with the M code which locks C axis in drilling

No.	Message	Contents
		canned cycle. Modify the program.
190	Servo spindle command occurs in rigid tapping	The increment of the servo spindle occurs in orientation command in rigid tapping. Modify the program.
197	C axis command error in spindle mode	When signal CON (G27#7) is OFF, CS contouring controlled axis is commanded to move. Modify the program or find the reason why the signal is not ON in ladder diagram.
198	Spindle speed arrival signal not found	During cutting feed, the spindle speed arrival signal SAR is not valid. Modify the program or check the ladder diagram.
201	Incorrect command used in macro program	An unusable function is specified in custom macro program. Modify the program.
202	Format error in macro program	There is a format error in <Formula>. Modify the program.
203	Illegal variable number is used in macro program.	A value not defined as variable number is designated in the custom macro. Modify the program.
204	Macro program call repetition	M98, G65 or G66 is called in G66 modal state in the same program. Modify the program.
205	Bracket nesting error	The number of bracket nesting level exceeds 5. Modify the program.
206	Illegal operation data	The argument of SQRT is a negative value; The arguments of BCD and BIN are negative values, or the BIN argument value cannot convert to correct BCD code. Modify the program.
207	Excessive macro program modal call	Macro call or macro program modal call nesting exceeds 4 levels. Modify the program.
208	Branch of macro program cannot used in DNC and MDI operation	Branch of macro program is used in DNC and MDI operation. Modify the program.
209	End statement absent	DO-END is not 1: 1; the END block contains other illegal command or the branch cannot be made to a location within the loop. Modify the program.
210	Limited authority	Argument assignment cannot be executed in MDI or DNC mode due to limited authority. Modify the program.
211	Illegal repetition number	Condition 1≤n≤3 is not fulfilled (n in Don). Modify the program.
212	NC statement and macro call statement coexist in the same block	NC statement and macro call statement are used mixedly. Modify the program.
213	Illegal macro sequence number	The defined sequence number in branch command is not within 1~99999, or they cannot by searched. Modify the program.
214	Illegal argument address	An unallowable address is specified in <argument>.

No.	Message	Contents
		Modify the program.
216	Illegal argument value	The argument value is erroneous or illegal. Modify the program.
217	Data error in logical operation command	The data in logical operation command OR, XOR, AND are negative values. Modify the program.
218	G67 modal call cancel is commanded	When G66 macro modal call is not specified, G67 modal call cancel is commanded. Check if it is necessary to write G66 command. Please refer to parameter 6000#0G67 and modify the program.
231	The axis commands of NC and PLC compete with each other	The axis commands of NC and PLC compete with each other. Modify the program or ladder diagram.
232	PLC controlled axis unchangeable	PLC axis selection has been made among the PLC controlled axes. Modify the Ladder Diagram.
251	The intersection point cannot be determined in tool nose radius compensation mode	The intersection point cannot be determined in tool nose radius compensation mode. Modify the program.
252	Tool nose radius compensation mode cannot be set or canceled in circular interpolation	Tool nose radius compensation mode is set or canceled in circular interpolation. Modify the program.
253	Compensation plane switching is not allowed in tool nose radius compensation	Compensation plane is changed in tool nose radius compensation mode. Modify the program.
254	Interference is generated in circular block in tool nose radius compensation mode	In tool nose radius compensation mode, the start point or end point of an arc is the same with the center point, or the end point is not on the arc, which may cause overcut. Modify the program.
255	In tool nose radius compensation mode, interference occurs in G90 or G94 block	Overcut may occur when tool nose radius compensation is commanded in G90 or G94 block. Modify the program.
256	Overcut occurs during interference check in tool nose radius compensation mode	The overcut may occur in tool nose radius compensation mode. Modify the program.
257	The cutter path direction is different with the programmed path direction in tool nose radius compensation mode.	The tool path direction is different with the programmed path direction in tool nose radius compensation mode (90°~270° difference). Overcut may occur. Modify the program.
258	G41 or G42 execution is not allowed in MDI mode	G41 or G42 (tool nose radius compensation) is specified in MDI mode. Please refer to parameter

No.	Message	Contents
		5008#4 MCR and modify the program.
259	Overcut is produced within the cutting full circle	Overcut is produced within the cutting full circle in tool nose radius compensation mode. Please refer to parameter 5008#5 CNF and modify the program.
260	Overcut may be produced when a step less than the tool radius is machined	In tool nose radius compensation mode, overcut may be produced when a step less than the tool radius is machined. Please refer to parameter 5008#6 CNS and modify the program.
261	The circular radius is less than the tool radius when a inner circle is machined	In tool nose radius compensation mode, overcut may occur if the circular radius is less than the tool radius when an inner circle is machined. Modify the program.
262	Circular command occurs when tool nose radius compensation is temporarily cancelled or set	In tool nose radius compensation mode, when G command for which the compensation mode needs to be temporarily cancelled is specified, circular command is specified to set or cancel the compensation mode. Modify the program.
263	Error is found in tool nose radius compensation mode	Programming error or operation error is found in tool nose radius compensation mode. Modify the program.
281	Illegal tool group number	The tool group number is larger than permitted. Modify the program.
282	Tool group number not found	The specified tool is not set. Modify the program or parameter.
283	Low capacity for tool storage	The tool number in one group exceeds the maximum amount. Modify the program.
284	T code not found	In tool lift management storage, T code is not stored. Modify the program.
285	P/L command not found	P and L commands are not at the head of the program which sets the tool group. Modify the program.
286	Tool many tool groups	The set tool group number exceeds the maximum amount. Modify the program.
287	Illegal tool life data	The tool life value is too large. Modify the setting value.
288	Tool data setting uncompleted	When the tool life data is setting, the power is turned off. Set the data again.

A.2 Parameter Alarms

No.	Message	Contents
400	Parameter writing is enabled	Press 【RESET】key to cancel the alarm.
401	The same servo communication property is set.	Please modify parameter No.9020.
402	Parameter backup failure	Please check the memory or re-power on.
403	Parameter recovery failure	Please check if the parameter is being writing, or re-power on and try it again.

No.	Message	Contents
404	The same axis name is set.	Modify the parameter NO.1020.
406	The slave numbers of non-Cs axis and spindle are set the same	Modify parameter No. 3704, No.8133,No.9020,No.9030.
407	The slave numbers Cs axis and spindle are inconsistent	Modify parameter No.3704,No.8133,No.9020,No.9030
408	The slave numbers of spindle is set the same	Modify parameter No. 9030
450	Parameter modification is done, please re-power on.	The input parameter is valid only after re-power-on.
452	The CNC controlled axis number is greater than the total axis number.	Please check parameter No. 1010 and 8130.
453	The same axis property is set.	Modify parameter No. 1022.
454	The same servo axis number is set.	Modify parameter No. 1023.
455	The rotary axis is conflicting with the axis property.	Parameter No.1006 conflicts with parameter No. 1022. The axis property of rotary axis cannot be non-0 value. Modify parameters No. 1006 or No. 1022.

A.3 Pulse Encoder Alarms

No.	Message	Contents
500	n-th axis origin return (n represents axis number)	Manual reference position return is required for the n-th axis.
501	The nth axis communication error	n-th axis absolute pulse encoder (APC)communication error; Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
502	The n-th axis overtime	n-th axis APC overtime error; Failure in data transmission. Possible causes include a faulty APC, cable or servo interface module.

A.4 Servo Alarms

No.	Message	Contents
604	n-th axis servo alarm	Faulty digital servo system of n-th axis
650	Power failure alarm	The power is turned off when the movement command is executed by the servo. It may lead to incorrect coordinate position. Please perform reference point return again.

A.5 Overtravel Alarms

No.	Message	Contents
700	n axis + side stroke limit 1	The positive stored stroke limit 1 has been exceeded. Modify the parameter No. 1320.
701	n axis - side stroke limit 1	The negative stored stroke limit 1 has been exceeded. Modify the parameter No. 1321.
702	n axis + side stroke limit 2	The positive stored stroke limit 2 has been exceeded. Modify the parameter No. 1322.
703	n axis - side stroke limit 2	The negative stored stroke limit 2 has been exceeded. Modify the parameter No. 1323.
704	n axis + side stroke limit 3	The positive stored stroke limit 3 has been exceeded. Modify the parameter No. 1324.
705	n axis - side stroke limit 3	The negative stored stroke limit 3 has been exceeded. Modify the parameter No. 1325.
706	Overtravel: +n	The positive hardware stroke limit has been exceeded. Please press overtravel release or modify parameter No. 3004.
707	Overtravel : -n	The negative hardware stroke limit has been exceeded. Please press overtravel release or modify parameter No. 3004.

A.6 Spindle Alarms

No.	Message	Contents
800	Spindle 1 alarm	Spindle 1 alarm occurs.
810	Spindle 2 alarm	Spindle 2 alarm occurs.

A.7 System Alarms

No.	Message	Contents
900	Memory alarm	Memory distribution error
909	The running time is up. The system cannot work normally	Please contact the sales personnel.
910	Initialized parameter error	Parameter file is not existed or the data is corrupted. The default configuration is used.
911	Initialized CNC configuration error	CNC configuration file is not existed or the data is corrupted. The default configuration is used.
912	Initialized tool compensation data error	The tool compensation file is not existed or the data is corrupted. The default data is used.
913	Initialized tool life data error	Tool life file is not existed or the data is corrupted. The default data is used.

914	Initialized pitch error compensation data error	The pitch error compensation data file is not existed or the data is corrupted. The default data is used.
915	Initialized PLC program error	File read failure or compilation error during loading
916	CNC initialization failure	Turn ON the power again
917	GSKLink initialization failure	Please check parameter No. 9000-No.9030 and the communication interface and power supply grounding state. Press RESET key to cancel the alarm or turn on the power again.
918	Editing keyboard or panel fault	Press 【RESET】key to cancel the alarm, or re-power on.
919	Memory fault	Remove the alarm by pressing RESET key. Please re-power on or resort to depot repair.
920	Too many alarms or prompts	The total number of alarms exceeds 14 or prompts exceed 20.
921	Unrecognized alarm number	There is no alarm content for the alarm number.
922	Data error in alarm information	In alarm information or operation information, some data are erroneous.
950	The pulse number inconsistent	The FPGA pulse number send from the system is inconsistent with the actual feedback pulse number.
998	Abnormal data	Please contact system developer.

A.8 PLC Alarms

Standard PLC Alarm (Address A) Instruction		
Address	**No.**	**Message**
A0000.0	1000	Tool change time is too long.
A0000.1	1001	When reverse rotation is completed, the current tool position is inconsistent with the expected one.
A0000.2	1002	Tool change uncompleted
A0000.3	1003	The tailstock function is disabled. M10/M11 command cannot be executed.
A0000.4	1004	Retracting from the tailstock is not allowed during spindle rotation.
A0000.5	1005	The spindle enabling function is closed. Spindle cannot be started.
A0000.6	1006	Protection door is not closed. Machining or spindle start is forbidden.
A0000.7	1007	Low pressure alarm
A0001.0	1008	The chuck cannot be released during spindle rotation
A0001.1	1009	The chuck is not clamping tightly, spindle cannot be started.
A0001.2	1010	Chuck clamping signal is not found during spindle rotation.

A0001.3	1011	The chuck is unclamped. Spindle start is forbidden.
A0001.4	1012	The chuck function is disabled. Command M12/M13 cannot be executed.
A0001.5	1013	Tool post locked signal is not found at the end of tool change.
A0001.6	1014	M code undefined
A0001.7	1015	Non-analog spindle; the spindle jog function cannot be executed.
A0002.0	1016	M03, M04 specification error
A0002.1	1017	Automatic gear changing is forbidden during spindle rotation.
A0002.2	1018	D0 setting error (D0 should be less than or equal to 8 and greater than 0)
A0002.3	1019	Non-analog spindle; command M41, M42, M43, M44 cannot be executed.
A0002.4	1020	Automatic gear changing is disabled. Check parameter K13.7
A0002.5	1021	Cycle start is not allowed at feeding hold knob position
A0002.7	1023	Short circuit detected on the machine panel
A0003.0	1024	The specified tool number is larger than the maximum number of tools (D0)
A0003.1	1025	Specified M code invalid
A0003.2	1026	Spindle orientation time is too long
A0003.3	1027	Chuck clamp/release in-position signal is not found
A0004.0	1032	Pre-indexing proximity switch signal is not received
A0004.1	1033	Lock proximity switch signal is not received
A0004.2	1034	The current tool number is inconsistent with the expected one when tool change is finished.
A0004.3	1035	No lock proximity signal when the tool change is finished.
A0004.4	1036	Tool post overheat
A0004.5	1037	D0 setting error (only 8, 10 and 12 are allowed)
A0005.0	1040	Expected tool number not found alarm
A0005.1	1041	Tool post rotation stop and lock signal not found
A0005.2	1042	No lock signal when tool change is finished.
A0005.3	1043	The current tool number is inconsistent with the expected one when tool change is finished.
A0005.4	1044	D0 setting error (only 8 is allowed)

A.9 GSKLink Communication Prompts

No.	Message	Contents	Possible Reason
5000	GSKLink slave configuration method error	GSKLink extended function unusable.	Unused at present
5001	I/O unit missing in GSKLink communication	The IO unit control function is unusable.	Unused at present
5002	Extended slave is missing in GSKLink communication	The extended axis function is unusable.	Unused at present
5003	communication error	Please check whether the communication interface is loose, the power supply is grounded properly, or the end resistance is installed, then, turn on the power.	During GSKLink communication, if error continuously occurs in all slaves, this prompt is displayed.
5004	GSKLink slave ID number conflicted	Modify the parameter for slave number and re-power on (cut off the GSKLink connection before parameter modification at the server side)	This prompt is displayed when two slave numbers of servos are set the same.
5005	All GSKLink slaves connections failure	Check the setting of parameter No.9000-No.9012 and check whether the communication interface is loose, the power supply is grounded properly or the end resistance is installed, then turn ON the power again.	When GSKLink is restarted or re-connected, all the slaves are cut OFF. The possible reasons are: (1). Poor contact of system GSKLink communication interface (2). Poor contact of servo slave GSKLink communication interface. (3). End resistor is not installed on the servo slave which is the farthest from the system. (4) GSKLink communication is interrupted. (5). Power supply is not grounded.
5006	n-th axis GSKLink slave connection failure	Check whether the communication interface is loose or the power supply is grounded properly.	The same as the prompt No. 5005, but this prompt indicates that only some slave connection is failed.
5010	n-th axis servo model and software version read failure	Check whether the communication interface is loose or the power supply is grounded properly, then turn ON the power again.	GSKLink communication is interrupted.

No.	Message	Contents	Possible Reason
5011	n-th axis servo configuration failure	Please update relevant servo configuration file and turn ON the power again.	The servo configuration file is not found, or the data in the file is unusable.
5020	n-th servo parameter read failure	Please check whether the communication interface is loose or the power supply is grounded	GSKLink communication is interrupted
5030	The parameter in the n-th axis current servo parameter file is inconsistent with the read one	Please select a valid servo parameter.	After the servo is disconnected with the system, servo parameter is manually changed on the drive unit. When this servo is used the next time after power-on, an alarm occurs. Note: When a servo of different version is used, the system will automatically select the parameters read in the servo, and an alarm will not occur.
5031	The parameter of the n-th production servo parameter is inconsistent with the read one	The parameter of the n-th production servo parameter is inconsistent with the read one (such as the encoder zero drift, drive unit version etc.) You could select the read servo parameter or other parameter stored in CNC servo parameter files.	This alarm occurs together with alarm No. 5030; It occurs only when some parameters (such as encoder zero drift, drive unit version) are inconsistent with the current stored parameters. These parameters includes two types: one can be modified manually after the communication is disconnected and logining in the drive unit; the other one can only be modified by upgrading servo software. When the two types are not consistent, the parameter read from the servo system should take priority and the stored value in CNC current parameter file should be overwritten.

A.10 Servo Inner Alarms

Note: (1). n represents the sequence number of GSKLink servo slaves set by system parameters (ranges from 1~9).

(2). The examples shown in the following table are feed servo V1.03 and spindle V2.02. Previous versions are compatible.

(3). The following content is valid till this user manual is issued and it is changed without further notice. Please refer to the latest servo manual.

Feed Servo	DAT2030C, DAT2050C, DAT2075C, DAT2100C (V1.03)	
No.	Message	Contents
5n00	Normal	
5n01	Overspeed	The speed of servo motor exceeds the setting value.
5n02	Overvoltage	The main voltage is too high.
5n03	Undervoltage	The main voltage is too low.
5n04	Excess position deviation	The position deviation value exceeds the setting value.
5n05	Overheat	The temperature of the motor is too high.
5n06	Speed amplifier saturated	The speed regulator is saturated for a long time.
5n07	Drive unit inhabitation abnormal	The drive unit input inhabitation is OFF.
5n08	Position deviation counter overflow	The absolute value of position deviation counter value exceeds $2^{30.}$
5n09	Coder fault	Coder signal error
5n10	Undervoltage of control power	The voltage of the control power is less than ±12V.
5n11	IPM module fault	IPM intelligent module fault
5n12	Overcurrent	The current of the motor is excessive.
5n13	Unused	
5n14	Braking fault	Braking circuit fault
5n15	Unused	
5n16	Motor overheat	The heat value of the motor exceeds the setting value. (I^2t detection)
5n17	Unused	
5n18	Unused	
5n19	Unused	
5n20	EEPROM error	(EEPROM) error
5n21	Phase lose alarm	Phase lose during the three-phase AC current input
5n22	Coder zeroing alarm	The encoder cannot perform normal regulation.
5n23	Current sampling circuit fault	A/D chip or current sensor error
5n24	Unused	
5n25	Unused	
5n27	Unused	
5n28	Software upgrade prompt alarm	The alarm is issued when the system software is upgrading.
5n29	Parameter error	The parameter is out of the controllable range.
5n30	Unused	
5n31	Unused	
5n32	illegal code in UVW signal	Full high-level or full low-level exists in UVW signal.

5n33	Power charging fault	Charging circuit is damaged.
5n34	Pulse electronic gear ratio is excessive	The parameter of pulse electronic gear ratio is incorrect.
5n35	No external connected brake pipe	There is no external connected brake pipe or the pipe is faulty.
5n36	Three-phase power OFF	Three-phase power OFF or three-phase power detection circuit is faulty.
5n37	The temperature of the radiator is too low	
5n38	The temperature of the radiator is too high	
5n39	Absolute encoder single-ring read alarm	
5n40	Absolute encoder multi-ring read alarm	
5n41	Encoder type configuration error	The encoder type set by drive unit is inconsistent with the encoder type of the motor.
5n42	EEPROM alarm in absolute encoder	
5n43	EEPROM check error in absolute encoder	
5n44	Coder type error	Please check parameter No. PA97.
5n45	Data check error in absolute encoder	Data check error in sensor mode.

Spindle Servo	**DAY3025C, DAY3100C, DAP03C**	
No.	**Message**	**Contents**
5n00	Normal	
5n01	Motor overspeed	The speed of the spindle motor exceeds the setting value.
5n02	Main circuit overvoltage	The voltage of the main circuit power is excessive.
5n03	Main circuit undervoltage	The voltage of the main circuit power is too low.
5n04	Excess position deviation	The position deviation value exceeds the setting value.
5n05	Motor overheat	The temperature of the motor is too high.
5n06	Unused	
5n07	Unused	
5n08	Position deviation counter overflow	The absolute value of position deviation counter value exceeds 2^30.
5n09	Motor encoder fault	The signal of motor encoder is faulty.
5n10	Unused	
5n11	IPM module fault	IPM intelligent module fault

5n12	Unused	
5n13	Overload	The current of the motor is excessive.
5n14	Unused	
5n15	Unused	
5n16	Motor overheat	The spindle servo drive unit and motor are overloaded (temporary overheat).
5n17	Excess braking time	This alarm is issued when the discharging time is too long.
5n18	Braking circuit fault 1	No braking signal, no braking feedback
5n19	Braking circuit fault 2	No braking signal, no braking feedback
5n20	EEPROM error	EEPROM error
5n21	Phase lose alarm	At least one of the R,S,T of three-phase power is off.
5n22	Unused	
5n23	Excessive current error	The zero drift is excessive.
5n24	Spindle encoding disc fault	The spindle encoder signal error
5n25	Orientation failure	The position cannot be found.
5n26	Cooling fins overheated	The cooling fins are overheated.
5n27	U, V, W connection error	The three-phase (U, V, W) sequence is wrong
5n28	The parameters are not re-adjusted or stored after upgrading	
5n29	The parameter value detected after power-on is out of the range	
5n30	Communication error	The connection between servo and CNC is faulty.
5n31	Unused	
5n32	Unused	
5n33	Charging alarm fault	The input voltage is less than 304V (DC bus voltage 430V).
5n34	Abnormal thermistor status	TEP-OH (TEM higher than 90°) or TEP-OL(TEP lower than -30°), the thermistor is short-circuited or cut off.

APPENDIX B MOTOR TYPE CODE LIST

B.1 DAT2000C Series Motor Model Code List

DAT2050C(5.03)			
Model code value	**Model**	**Model code value**	**Model**
PA001=0	130SJT-M075D(A4)	PA001=3	130SJT-M100D(A4S)
PA001=1	130SJT-M075D(A4S)	PA001=4	130SJT-M150D(A4)
PA001=2	130SJT-M100D(A4)	PA001=5	130SJT-M150D(A4S)

DAT2075C(5.03)			
Model code value	**Model**	**Model code value**	**Model**
PA001=0	130SJT-M075D(A4)	PA001=3	130SJT-M100D(A4S)
PA001=1	130SJT-M075D(A4S)	PA001=4	175SJT-M300D(A4)
PA001=2	130SJT-M100D(A4)	PA001=5	175SJT-M300D(A4S)

DAT2000C(V1.03)			
Model code value	**Model**	**Model code value**	**Model**
PA001=0	Reserved	PA001=43	130SJT-M050D(A4SI)
PA001=1	Reserved	PA001=44	130SJT-M060D(A4I)
PA001=2	Reserved	PA001=45	130SJT-M060D(A4SI)
PA001=3	Reserved	PA001=46	130SJT-M075D(A4I)
PA001=4	80SJT-M024C(A4I)	PA001=47	130SJT-M075D(A4SI)
PA001=5	80SJT-M024C(A4SI)	PA001=48	130SJT-M100B(A4I)
PA001=6	80SJT-M024E(A4I)	PA001=49	130SJT-M100B(A4SI)
PA001=7	80SJT-M024E(A4SI)	PA001=50	130SJT-M100D(A4I)
PA001=8	80SJT-M032C(A4I)	PA001=51	130SJT-M100D(A4SI)
PA001=9	80SJT-M032C(A4SI)	PA001=52	130SJT-M150B(A4I)
PA001=10	80SJT-M032E(A4I)	PA001=53	130SJT-M150B(A4SI)
PA001=11	80SJT-M032E(A4SI)	PA001=54	130SJT-M150D(A4I)
PA001=12	Reserved	PA001=55	130SJT-M150D(A4SI)
PA001=13	Reserved	PA001=56	Reserved
PA001=14	Reserved	PA001=57	Reserved
PA001=15	Reserved	PA001=58	Reserved
PA001=16	Reserved	PA001=59	Reserved
PA001=17	Reserved	PA001=60	Reserved
PA001=18	Reserved	PA001=61	Reserved
PA001=19	Reserved	PA001=62	Reserved
PA001=20	110SJT-M020E(A4I)	PA001=63	Reserved
PA001=21	110SJT-M020E(A4SI)	PA001=64	Reserved

PA001=22	110SJT-M040D(A4I)	PA001=65	Reserved
PA001=23	110SJT-M040D(A4SI)	PA001=66	Reserved
PA001=24	110SJT-M040E(A4I)	PA001=67	Reserved
PA001=25	110SJT-M040E(A4SI)	PA001=68	175SJT-M150D(A4I)
PA001=26	110SJT-M060D(A4I)	PA001=69	175SJT-M150D(A4SI)
PA001=27	110SJT-M060D(A4SI)	PA001=70	175SJT-M180B(A4I)
PA001=28	110SJT-M060E(A4I)	PA001=71	175SJT-M180B(A4SI)
PA001=29	110SJT-M060E(A4SI)	PA001=72	175SJT-M180D(A4I)
PA001=30	Reserved	PA001=73	175SJT-M180D(A4SI)
PA001=31	Reserved	PA001=74	175SJT-M220B(A4I)
PA001=32	Reserved	PA001=75	175SJT-M220B(A4SI)
PA001=33	Reserved	PA001=76	175SJT-M220D(A4I)
PA001=34	Reserved	PA001=77	175SJT-M220D(A4SI)
PA001=35	Reserved	PA001=78	175SJT-M300B(A4I)
PA001=36	Reserved	PA001=79	175SJT-M300B(A4SI)
PA001=37	Reserved	PA001=80	175SJT-M300D(A4I)
PA001=38	Reserved	PA001=81	175SJT-M300D(A4SI)
PA001=39	Reserved	PA001=82	175SJT-M380B(A4I)
PA001=40	130SJT-M040D(A4I)	PA001=83	175SJT-M380B(A4SI)
PA001=41	130SJT-M040D(A4SI)	PA001=84	Reserved
PA001=42	130SJT-M050D(A4I)	PA001=85	Reserved

B.2 DAP03C,DAY3025C Model Code List

DAP03C,DAY3025C			
Model code value	**Model**	**Model code value**	**Model**
PA001=0	GM7101-4SB61，3.7kW	PA001=16	ZJY265-7.5A-B3(21.0A)
PA001=1	GM7103-4SB61，5.5kW	PA001=17	ZJY182-1.5B-B35(7.3A)
PA001=2	GM7105-4SB61，7.5kW	PA001=18	ZJY182-2.2B-B35(7.5A)
PA001=3	GM7131-4SB61，11kW	PA001=19	ZJY182-3.7B-B35(15.5A)
PA001=4	GM7103-4SC61，7.5kW	PA001=20	ZJY208-2.2B-B5(B3)(6.3A),
PA001=5	GM7100-4SB61，2.2kW	PA001=21	ZJY208-2.2B-B5(B3)(9.3A)
PA001=6	GM7109-4SB61，11kW	PA001=22	ZJY208-3.7B-B5(B3)(9.1A)
PA001=9	YPNC-50-2.2-B，2.2kW	PA001=23	ZJY208-5.5B-B5(B3)(13.2A)
PA001=10	YPNC-50-3.7-B，3.7kW	PA001=24	ZJY208-7.5B-B5(B3)(17.3A)
PA001=11	YPNC-50-5.5-B，5.5kW	PA001=25	ZJY265-7.5B-B5(B3)(18A)
PA001=12	YPNC-50-7.5-B，7.5kW	PA001=26	ZJY265-11B-B5(B3)(26A)
PA001=13	YPNC-50-11-B，11kW	PA001=27	ZJY265-15B-B5(B3)
PA001=14	YPNC-50-15-B，15kW	PA001=28	ZJY265-15A-B5(B3)(48.3A)

APPENDIX C COMMON ALARM REMEDY

C.1 CNC Common Alarm Remedy

No.	Meaning	Possible Reason	Remedy
000	Emergency stop, ESP open circuit	Restore the ESP emergency stop signal input to cancel the alarm.	000
001	Part program open failure	Program is not downloaded before the running in AUTO mode.	Download the program to be executed.
400	Parameter writing enabled	Parameter writing is enabled.	Press RESET key to cancel the alarm
450	Parameter modification finished, turn ON the power again	Parameter is modified and is only valid after power-on again.	Turn ON the power again.
500	The nth axis reference point return	When the feed axis whose motor adopts absolute encoder did not perform manual reference point operation after power-on.	Execute reference point return for corresponding axis.
604	The nth axis servo alarm	Digital servo system fault	Check and cancel servo alarm.
917	GSKLink initialization failure	1. The servo slave number set by system parameter is inconsistent with the one set in servo system.	Modify the parameter and check the communication line status.
		2. GSKLink Communication line is connected improperly.	

C.2 DAT 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.

No.	Meaning	Main Reason	Remedy
		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.
		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.
		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.

No.	Meaning	Main Reason	Remedy
		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-14	Braking circuit fault	1. The braking circuit is low in capacity.	A. Reduce the load. B. Change to a new drive device of higher power. C, Lower down the braking frequency.
		2. The inner braking circuit is damaged.	Change to a new drive unit.
		3. Braking resistor is cut off.	Re-connect the braking resistor line.
Err-16	Motor thermal overheat	1. The rated current parameter is set incorrectly.	Set the parameter according to the motor nameplate.
		2. The motor is running with excess current for a long time.	A. Reduce the load. B. Change to a drive device and motor of higher power. C Check whether the mechanical part is abnormal.
Err-20	When the power is ON, EEPROM alarm occurs in the	1. When the power is ON, the drive unit fails to read the data in EEPROM.	Recover the motor default parameter.

No.	Meaning	Main Reason	Remedy
	inner driver unit.	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-23	Current sampling error	1. The current sensor's working voltage is abnormal or the device is damaged.	Change the drive unit.
		2. Current sampling circuit resistor is damaged.	
Err-25	Power failure alarm	1. The main power is cut OFF after it is ON.	Check the power supply line.
		2. The rectification part of the drive unit is damaged.	Change the drive unit.
Err-32	Illegal code of encoder signals U,V,W	1. PA48 parameter setting is erroneous.	According to the matched encoder type, set the PA48 correctly and adjust to the default setting.
		2. The interface is poorly contacted or the cable is poorly shielded.	Check the encoder interface and shielding line.
		3. Encoder U,V,W signals are damaged.	Chang a new encoder.
		5. Encoder interface circuit fault.	Change to a new drive unit.
Err-33	Power charging fault	The charging circuit is damaged.	Change the drive unit.
Err-34	Pulse electronic gear ratio	The parameter setting of pulse electronic gear ratio is irrational.	Set the PA12/PA13 correctly.
Err-35	Alarm for the absence of external brake pipe	The external brake pipe is loose, or the external brake pipe is faulty.	Re-connect the brake pipe, or change the brake pipe.
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 -30℃.	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.

No.	Meaning	Main Reason	Remedy
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.
		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-41	Encoder type configuration error	The encoder type set by the drive unit is inconsistent with the actual type.	Change the encoder or change the encoder type of drive unit.
Err-42	EEPROM error read in 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.
		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.

C.3 Spindle Servo Alarm Remedy

No.	Meaning	Main Reason	Remedy
Err-1	The spindle motor speed exceeds the setting value (refer to the upper limit set by parameter PA23)	1. Encoder feedback signal is abnormal.	Check the motor or the second position encoder and its signal connection status.
		2. In speed mode, the acceleration/deceleration time constant is set too small, which lead to excessive speed overshoot.	Increase the accelerated time (PA39) or the decelerated time (PA40).
		3. Setting value of PA23 (maximum speed limit) is too small or the setting value of PA49 (motor encoder line number) is smaller than the actual encoder line number.	Set the PA23, PA49 value according to the motor nameplate.
		4, Control board is faulty.	Change the drive unit.
Err-2	The DC bus voltage of main circuit is too high	1. The brake resistor is disconnected or damaged.	Check brake resistor and its connection.
		2. The brake resistor is not matched (resistance is too large) Note: The smaller the brake resistance is, the larger the current of the brake circuit is, which will easily cause damage to the brake pipe.	A. Change to a brake motor whose resistor is matched with the power. B. Reduce the start/stop frequency according to the usage. C. Adjust the acceleration/ deceleration time and speed mode according to the usage.
		3. The power voltage is instable.	Check the power supply.
		4. The internal brake circuit is damaged.	Change the drive unit.
Err-3	The DC bus voltage of main circuit is too low	1. The input voltage capacity is not enough, which leads to the low voltage.	Check the power capacity and electric part of the control cabinet.
		2. If it occurs when the power is ON, it means the drive unit control board is faulty.	Change the drive unit.
Err-4	The value of position deviation counter exceeds the setting value (refer to the	1. The pulse command frequency is too high or the set electronic gear ratio is too large.	Check the command frequency on principal computer and the electronic gear ratio set by PA12/PA13.

No.	Meaning	Main Reason	Remedy
	position deviation detection range set by PA17) (PA18=0: detects the alarm of position deviation; PA18=1 : does not detect the alarm of position deviation)	2. The load inertial is excessive, or the torque is insufficient.	A. Check the overload magnification setting of the motor. (refer to PA34) B. Increase the power of the drive unit and the motor. C. Reduce the load.
		3. The motor encoder is faulty or the encoder line number setting is erroneous.	Check the motor encoder and its connection status; check the setting of PA49.
		4. The U, V, W phase sequence is erroneous, and	Exchange any two phases.
		5. When the second position encoder is used, PA68 is set incorrectly; the feedback signal is abnormal.	Check the setting of PA68.
		6. The position loop or speed loop gain setting is too small. (refer to PA5, PA6, PA9);	Adjust the speed loop or position loop gain.
		7. The valid range of position deviation is set too small.	Set PA17 correctly.
Err-5	Motor overheat alarm; the drive unit detects the overheat alarm signal output by the motor. (PA73=0: detects the motor overheat alarm; PA73=1: does not detects the overheat alarm)	1. There is no temperature detection device in the motor.	Set PA73 to 1 to shield the motor overheat alarm.
		2. Overload leads to severe heat of the motor.	Increase the power of drive unit and motor or reduce the load.
		3. In overload condition, the start/stop frequency is too high.	Reduce the start/stop frequency, and improve the motor heat radiation condition.
		4. The motor temperature detection device is damaged, or the motor inner is faulty, or the cooling fan is damaged.	Change the spindle servo motor.
		5. If the temperature detection signal is normal, the drive unit control board is faulty.	Change the drive unit.
Err-8	Position deviation counter overflow	1. The electronic gear ratio is too large.	Check the setting of parameters PA12, PA13.
		2. Input command pulse is abnormal.	Check the principal command pulse frequency.
Err-9	The encode signal feedback is abnormal.	1. The motor encoder signal is poorly connected or the connection is erroneous.	Check the connector and signal line welding status.
		2. The motor encoder signal feedback cable is too long, leading to low signal voltage.	Shorten the cable length within 30m.

No.	Meaning	Main Reason	Remedy
		3. Motor encoder is damaged.	Change the motor or the encoder.
		4. Drive unit control board is faulty.	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.	The remedy for reason A is to change the motor line or the motor. The remedy for reasons B is to change the drive unit.
		4. It occurs when the motor is started or stopped, and can be removed after power-on. A. The default parameter set by the drive unit is erroneous. B. The load inertia is too large; the command accelerated speed during start/stop is too large.	The remedy for reason A is to recover the motor default parameter. (refer to 4.4 section for the procedures); The remedy for reason B is to increase the acceleration/ deceleration time so as to reduce the command speed rate or the load inertia.
Err-13	Overload alarm when the motor is running	1. long time overcurrent;	Reduce the load.
		2. The parameter is set incorrectly. The motor may be accompanied with vibration or noise.	Adjust the parameter related to the motor performance (refer to the instruction to PA5, PA6, PA8, PA9, PA34)
		3. The setting value of PA49 is larger than the actual encoder line number.	Set the line number correctly.

No.	Meaning	Main Reason	Remedy
		4. Lines for U, V, W are connected incorrectly. The status after power-on is similar to the description in Err-27.	Exchange any two phases.
Err-16	Overload alarm when the motor is running	The motor is running in overload state for a long time (longer than the time in Err-13).	A. Reduce the load. B. Change to a drive device with higher power.
Err-17	Excessive braking time	1. The voltage of input power is too high for a long time.	Supply with the power which meets the need of drive unit working condition.
		2. The braking resistance is too large, and the energy cannot be released during braking, leading to the rise of inner DC voltage.	Change to a new braking resistor (refer to section 1.4.3);
Err-18	No feedback for the braking start signal	1. The brake circuit is faulty.	Change the drive unit.
		2. The braking resistance is too large.	Measure the resistance with a universal meter and adjust the resistance.
Err-19	Excessive DC bus voltage without braking	1. Braking circuit is faulty.	Change the drive unit.
		2. The braking resistor is loose or disconnected.	Check the connection status of the braking resistor.
Err-20	EEPROM alarm when the power is ON	1. When the power is ON, the drive unit fails to read the data in EEPROM.	Recover the motor default parameter. Refer to section 4.4 for the procedures.
		2. EEPROM chip or circuit is faulty.	Change the servo drive unit.
Err-21	R, S, T open phase alarm	1. The input power line is disconnected or the power is open-phase	A. Check the input power line; B. Check the input three-phase power.
		2. The input power circuit of drive unit is faulty.	Change the drive unit.
Err-23	Excessive current error	1. The current detection circuit is faulty.	Change the drive unit.
		2. The current sensor is damaged.	
		3. The voltage of control power supply is faulty.	
Err-24	The second position input signal of CN3 interface is abnormal.	1. Parameter PA66 is set to 1 when no second position feedback signal is received.	Set the PA66 to 0.
		2. The spindle encoder feedback signal is abnormal. (The reason is the same as described in Err-9)	A. Check the second position encoder signal connection line, welding status and plug connection status. B. Shorten the cable within 30m.

No.	Meaning	Main Reason	Remedy
Err-25	Drive unit positioning failure	1. No Z pulse signal is detected.	Check the connection of feedback input signal.
		2. Because of excess load inertia, the corresponding parameter is set incorrectly or the gain setting is too large.	Check relevant motor parameter values PA49, PA66, PA67. Relevant gain parameter PA5, PA6, PA8, PA9. (Refer to section 6.1 for the adjustment method.)
		3. When the positioning is performed with the second position input signal, the A/B phase sequence of spindle encoder is inconsistent with that of the motor encoder.	Modify parameter PA68 and reconcile the phase. Refer to instruction of parameter PA68 for details.
Err-26	Drive unit radiator overheat alarm	1. The radiator temperature is too high or the radiator fan is damaged.	Turn OFF the power and start the motor after cooling down. Check the radiator fan, clean the radiation passage and reduce the load.
		2. The temperature detection switch or circuit is damaged.	Change the drive unit.
Err-27	U, V, W connection error	The phase sequence of U, V, W is incorrect.	Exchange any two phases.
Err-28	Parameter for updating software erroneous	Parameters are not adjusted or saved after programming or software updating.	Re-call the default parameter and save them, then, turn on the power again.
Err-29	Parameter detection error after power-on	Conflict arises when the old version software is replaced by the new one.	Re-writer the parameter and turn ON the power again.
Err-33	Voltage of the main circuit is abnormal when the power is ON	1. At the power-on moment, the input power voltage is too low or it fluctuates greatly.	Check the input power.
		2. The rectifier is damaged or the soft-start circuit is faulty.	Change the drive unit.
Err-34	The temperature of radiating fin is abnormal. (Applies to DAY3025, DAY3100)	1. The temperature of the radiating fin is out of the range -30℃～90℃	Lower down the temperature.
		2. The thermistor is abnormal.	Change the drive unit.