



In this manual, we will endeavor to describe various matters related to the operation of the CNC System developed and manufactured by GSK CNC Equipment Co., Ltd.

It is not possible to describe in detail all the operations that need not be done and/or cannot be done in the system due to length constraints and specific product usage. Therefore anything not specifically indicated in this Instruction Manual shall be deemed to be the operation that is "impossible" or "not permitted".



The copyright of this Instruction Manual belongs to GSK CNC Equipment Co., Ltd., and it is illegal for any units or individuals to publish or copy it. GSK CNC Equipment Co., Ltd. reserves the right to investigate their legal liabilities.

Preface

Dear Customers:

We are honored and grateful to you for choosing CNC system from GSK CNC Equipment Co., Ltd.

This Manual is the "Installation and Commissioning Manual" (software version: V2.22) of the CNC System, which details the installation and commissioning of the CNC System.

Please read this Manual carefully before installing and using the series products in order to ensure product safety and normal and valid operation.

Safety Warnings



Improper operation would lead to accidents, and personnels with appropriate qualifications can be up to operating the system.

Special Tip:

The power supply installed on (in) the chassis is specially provided for the CNC Systems manufactured by our company.

Users are not allowed to use this power supply for other purposes. Otherwise, great danger will be caused!

Important Notice

■ Transport and storage

- Product packing boxes shall not be stacked for more than six layers.
- Do not climb, stand or place heavy objects on the packing boxes.
- Do not use cables connected to the product to drag or carry the products.
- Do not collide or scratch the panel and display screen.
- Product packing boxes shall be protected from dampness, exposure to the sun and rain.

■ Unpacking inspection

- Please make sure if it is the product you purchased after opening the package.
- Check whether the product is damaged in transit.
- Confirm if the components are complete and damaged according to the list.
- Please contact our company in time in case of product model inconsistency, missing accessories or transport damage.

■ Wiring

- Participants in wiring and inspection must be competent professionals.
- The product must be grounded reliably, with the grounding resistance no more than $0.1\ \Omega$. Neutral wire shall not be used in the placement of grounding wire.
- The wiring must be correct and firm to avoid product failure or unexpected consequences.
- Surge absorbing diodes must be connected to the product in the specified direction, otherwise the product will be damaged.
- The product must be powered off before plugging unplugging or opening the product chassis.

■ Overhaul

- Power supply must be cut off before repairing or replacing components
- The failure must be checked in case of short circuit or overload, and it can be restarted after troubleshooting
- The power to the product shall not be frequently turned on or off. The interval shall be at least 1 min if re-energization is required after power-off

Statement

- This Manual describes various contents as much as possible. However, due to too many possibilities involved, it is impossible to describe all the operations that can or cannot be performed one by one. Therefore, contents with no special description in this manual can be considered unusable.

Warnings

- Before programming and operating the product, you must read the Product Manual and the Operation Instructions of the machine tool manufacturer in detail, and perform related operations in strict accordance with the requirements of the Manual and the Instructions; otherwise it may cause damage to the product and machine tool, work piece scrapping or even personal injury.

Note

- The product functions and technical indicators (such as accuracy, speed, etc.) described in this manual are only for this product. The actual function configuration and technical performance of CNC machine tool are determined by the design of the machine tool manufacturer, which are subject to the instruction manual of the machine tool manufacturer.
- Please refer to the Operation Instructions of the machine tool manufacturer for the function and meaning of keys on the machine panel.

The content of this manual is subject to change without notice!

Safety Warnings

Manufacturer's Responsibility for Safety

- The manufacturer shall be responsible for the hazard of the supplied CNC System and accompanying accessories that have been eliminated and/or controlled in design and structure.
- The manufacturer shall be responsible for the safety of the supplied CNC System and accompanying accessories.
- The manufacturer shall be responsible for the use information and advice provided for the user.

Users' Responsibilities for Safety

- Users shall be familiar with and master the safe operation through learning and training of safe operation of CNC System.
- Users shall be responsible for the safety and hazards as a result of adding, changing or modifying the original CNC System and accessories by themselves.
- Users shall be responsible for any hazard caused by failure to operate, adjust, maintain, install, store and transport the products in accordance with the Manual.

This manual shall be kept by final users.

**Thank you very much for using the products of GSK CNC Equipment Co.,
Ltd. and for your friendly support to our company!**

Contents

Volume 1 Installment and Connection

Chapter 1 Interface Signal Definition and Connection	3
1.1 System Appearance.....	3
1.2 Control system.....	4
1.2.1 CNC back cover interface layout and its connection diagram.....	4
1.2.2 Machine operation panel back cover interface and its connection diagram	4
1.3 Product installation	5
1.3.1 Installation conditions of electrical cabinet	5
1.3.2 System grounding requirements	5
1.3.3 Methods to prevent interference	5
1.4 CNC host	7
1.4.1 GSK-Link bus interface.....	7
1.4.2 Spindle encoder interface.....	7
1.4.3 High-speed input interface.....	9
1.4.4 Communication interface.....	9
1.4.5 Power socket.....	10
1.4.6 Network port.....	10
1.4.7 Panel USB interface.....	11
1.5 CNC operation panel	11
1.5.1 Band switch dedicated interface	11
1.5.2 Dedicated interface for external key	13
1.5.3 MPG interface.....	14
1.5.4 Communication interface.....	15
1.5.5 General input/output address	15
1.5.6 Machine panel power interface.....	15
1.6 I / O unit.....	15
1.6.1 Bus interface CN5/CN5.....	15
1.6.2 Communication interface CN52	16
1.6.3 Spindle CN41 and spindle CN42.....	16
1.6.3.1 IOR-44T and IOR-44F.....	16
1.6.3.2 IOR-21F.....	16
1.6.4 Power interface CN1.....	17
1.6.5 Spindle Interface (Analog Output: 0~10V)	17
1.6.6 Pulse Axis Interface (IOR-20TP and IOR-23TP)	17
1.7 Use of CNC input and output signals	18
1.7.1 Input signal	18
1.7.2 Output signal.....	19
1.8 Use of I/O unit input and output signal.....	21
1.8.1 Input signal	21
1.8.2 Output signal.....	21

1.8.2.1	Output signal is low level	21
1.8.2.2	Output signal is high.....	21

Volume 2 Debugging

Chapter 1 Machine Tool Debugging and Operation.....27

1.1	Parameter setting	27
1.1.1	System parameters	27
1.1.2	Servo parameters	29
1.1.3	I/O unit parameters	31
1.2	U-disk operation	33
1.2.1	File management page	33
1.2.2	Program page	34
1.2.3	Ladder diagram page	36
1.3	PLC debugging operation	36
1.3.1	PLC run and stop	37
1.3.2	PLC monitoring and diagnosis	38
1.3.3	PLC data viewing and setting	42
1.3.4	PLC online editing	45
1.3.5	PLC program transmission	54
1.4	System debugging operation.....	55
1.4.1	Preparation for commissioning.....	56
1.4.2	Basic positions	57
1.4.3	Bus settings.....	63
1.4.4	Gear ratio setting.....	66
1.4.5	Reference point setting	68
1.4.5.1	Set reference point	68
1.4.5.2	Set travel limit.....	69
1.4.6	One-key backup/recovery function.....	70
1.4.6.1	One-key backup	70
1.4.6.2	One-key recovery.....	71
1.4.6.3	Delete configuration	72
1.5	Servo debugging operation	73
1.5.1	Servo adjustment	73
1.5.2	Roundness test	74
1.5.3	Biaxial error	76
1.5.4	Servo tuning.....	76
1.5.5	Encoder Calibration	77
1.6	Diagnostic information	77
1.6.1	System diagnosis	77
1.6.1.1	Edit keyboard diagnostics	77
1.6.1.2	Hardware interface diagnosis.....	78
1.6.1.3	Bus status diagnosis	79
1.6.1.4	Communication data diagnosis.....	79
1.6.2	Bus connection diagnosis.....	80

1.6.3	I/O diagnosis	81
1.7	Use of GSKUI software	82
1.7.1	Software overview	82
1.7.2	Usage method	83
1.7.3	Explanation of project directory	88
1.7.3.1	Explanation of project path	88
1.7.3.2	Directory structure explanation	88
1.7.4	File transfer	89
1.7.4.1	USB flash drive transfer	89
1.7.4.2	Network transfer	91
Chapter 2	Machine Debugging and Function.....	93
2.1	GSK-Link bus connection	93
2.2	Emergency stop and hard limit	95
2.3	Basic axis parameter setting	96
2.4	Gear ratio calculation and setting	98
2.4.1	Gear ratio calculation	99
2.4.2	Gear ratio setting	100
2.5	Servo related settings and adjustments	100
2.5.1	CNC servo parameter setting	100
2.6	Adjustment of acceleration and deceleration	101
2.7	Reference point and soft limit	104
2.7.1	Reference point setting of absolute encoder	106
2.7.2	Setting of reference point with stop	106
2.7.3	Setting of reference point without stopper	108
2.7.4	Stored stroke check settings	109
2.8	Error compensation function	111
2.8.1	Stored pitch error compensation	111
2.8.2	Double-directional pitch error compensation	117
2.8.3	Backlash compensation	119
2.9	Tapping function	121
2.9.1	Tapping method selection	121
2.9.2	Rigid tapping debugging settings	123
2.10	Function of spindle	128
2.10.1	Spindle encoder selection	128
2.10.2	Spindle speed control	129
2.11	Position switch function	134
2.12	Parallel program function	134
2.12.1	Parallel program editing	135
2.12.2	Parallel program execution	135
2.12.2.1	Parallel program content is determined by parameters	135
2.12.2.2	Content of parallel program is determined by main program	136
2.12.3	Parallel program display	137
2.12.4	Parallel program drilling, boring and rigid tapping	138
2.12.5	Precautions for drilling/boring cycles	139
2.12.6	Precautions for tapping cycle	140

2.13	Indexing function of indexing table.....	140
2.13.1	Function setting	141
2.13.2	Action steps for indexing table positioning	141
2.14	Additional axis function	143
2.14.1	Basic parameter setting of additional axis	144
2.14.1.1	Additional axis name	144
2.14.1.2	Type setting of additional axis	145
2.14.2	Calculation and setting of additional axis gear ratio.....	145
2.14.3	Servo parameter setting of additional axis.....	145
2.14.4	Additional axis acceleration/deceleration feature adjustment.....	146
2.14.5	Additional axis reference point and soft limit.....	147
2.14.5.1	Additional axis reference point setting	147
2.14.5.2	Additional axis soft limit setting.....	149
2.14.6	Additional axis position switch function	150

Chapter 3 Network Communication Connection..... 151

3.1	Network communication setting.....	151
3.2	FTP communication setting	151
3.3	Remote data communication function	152
3.4	MODBUS data communication function.....	152

Appendix

Appendix I Parameter Description 155

1.1	Parameters related to system settings	156
1.2	Parameters related to input and output interfaces.....	156
1.3	Parameters related to axis control/setting unit.....	158
1.4	Parameters related to torque control	163
1.5	Parameters about the coordinate system.....	166
1.6	Parameters related to stroke detection	170
1.7	Parameters related to infeed speed.....	175
1.8	Parameters related to acceleration and deceleration control.....	183
1.9	Parameters Related to servo and backlash compensation.....	187
1.10	parameters related to input and output.....	192
1.11	Parameters related to display and editing	200
1.12	Parameters related to programming.....	206
1.13	Parallel program.....	211
1.14	Parameters related to pitch error compensation.....	218
1.15	Parameters related to spindle control.....	221
1.16	Parameters related to tool compensation	236
1.17	Parameters related to the canned cycle	247
1.17.1	Parameters related to drilling canned cycles.....	247
1.17.2	Parameters related to thread cutting cycle	250
1.17.3	Parameters related to the compound canned cycle	251
1.18	Parameters related to rigid tapping	256

1.19	Parameters related to special processing.....	261
1.20	Parameters related to polar coordinate interpolation	264
1.21	Parameters about indexing table	267
1.22	Parameters related to user macro program.....	274
1.23	Parameters related to jump function	281
1.24	Parameters related to MPG rollback.....	284
1.25	Parameters related to graphic display	286
1.26	Parameters related to display of running time and number of parts	287
1.27	Parameters related to tool life management.....	288
1.28	Parameters on position switch	293
1.29	Parameters related to MPG feed	297
1.30	Parameters related to polygon processing.....	302
1.31	Parameters about electronic gearbox	305
1.32	Parameters related to PLC Axis Control.....	308
1.33	Parameters related to basic functions.....	312
1.34	Parameters related to inclined axis control	314
1.35	Parameters related to synchronous control of feed axis.....	316
1.36	Parameters related to additional axis	317
1.37	Parameters related to GSK-Link communication function.....	335
Appendix II Installation Layout		337
2.1	Button Structure (10.4 -inch).....	337
2.2	Button Structure (15-inch).....	338
2.3	I/O Unit Models.....	339
2.4	Vertical Button Structure Installation Dimension Diagram (10.4-inch)	340
2.5	Horizontal Button Structure Installation Dimension Diagram (10.4-inch).....	341
2.6	Vertical Button Structure Installation Dimension Diagram (15-inch)	342
2.7	I/O Unit Overall Dimensions	344
2.7.1	Installation Method and Overall Dimensions of IOR-04T/IOR-44T.....	344
2.7.2	Installation Method and Overall Dimensions of IOR-44F.....	344
2.7.3	Installation Method and Overall Dimensions of MCT07 (Compatible with IOR-21F/IOR-44F)	345
Appendix III Standard Ladder Diagram Function Configuration		347
3.1	The definition of the key address of the machine tool panel	347
3.2	The definition of standard ladder diagram X and Y address.....	348
3.2.1	High-speed I/O interface.....	348
3.2.2	General Machine Tool I/O Interface	348
3.2.3	Handheld Box Interface	352
Appendix IV Alarm Handling.....		355
4.1	CNC Alarm Handling	355

Volume 1 Installment and Connection

Chapter 1 Interface Signal Definition and Connection

1.1 System Appearance

The system is equipped with either a 10.4-inch LCD display or a 15-inch LCD display, along with independent keys. The 10.4-inch display offers two optional mounting structures: horizontal and vertical.

Figure 1-1 shows the external view of the 15-inch LCD display with independent keys in a vertical mounting configuration.

Figure 1-2 presents the external view of the 10.4-inch LCD display with independent keys in a vertical mounting configuration.

Figure 1-3 displays the external view of the 10.4-inch LCD display with independent keys in a horizontal mounting structure.

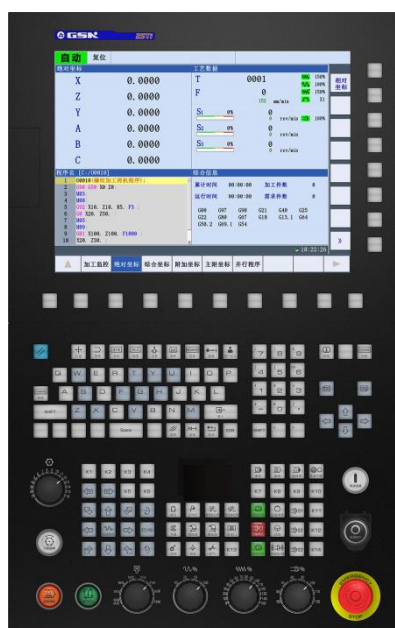


Fig. 1-1



Fig. 1-2



Fig. 1-3

1.2 Control system

1.2.1 CNC back cover interface layout and its connection diagram

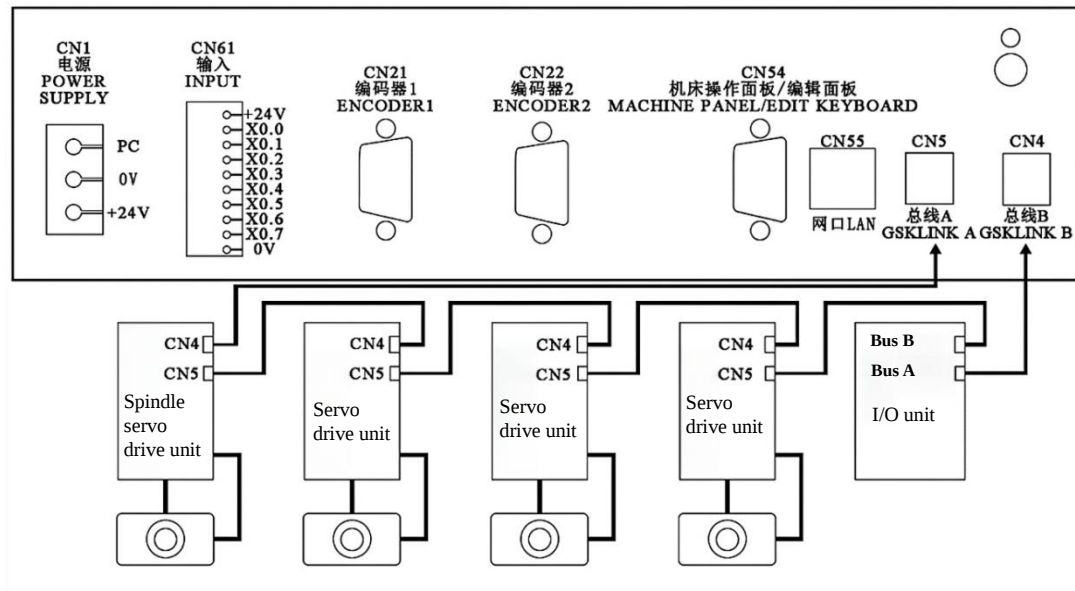


Fig. 1-4

1.2.2 Machine operation panel back cover interface and its connection diagram

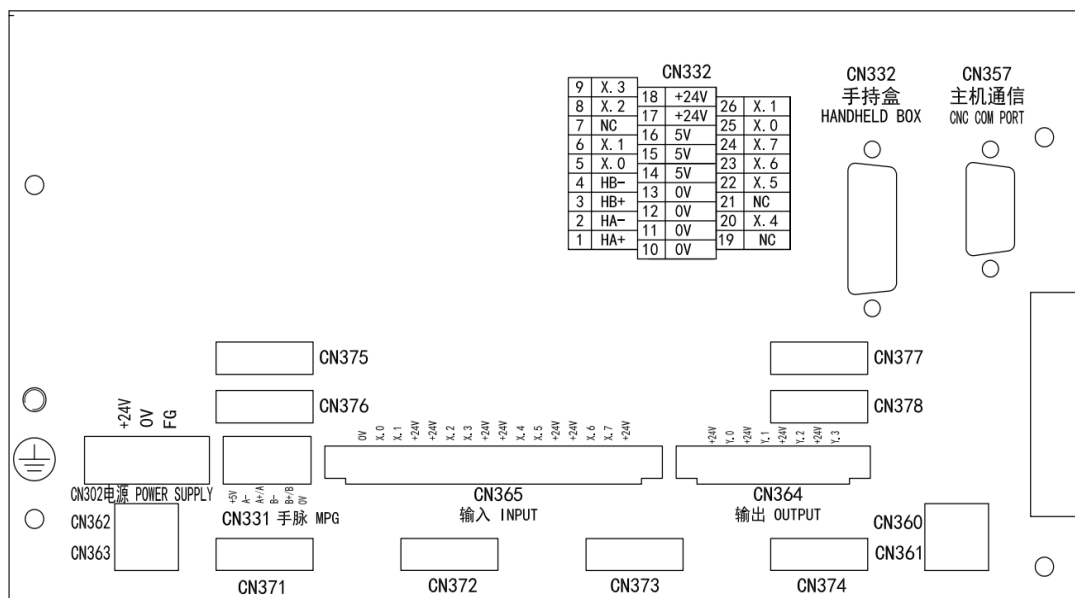


Fig. 1-5

1.3 Product installation

1.3.1 Installation conditions of electrical cabinet

- The electric cabinet must be able to effectively prevent the entry of dust, coolant and organic solution.
- When designing the electrical cabinet, the distance between the CNC back cover and the chassis should not be less than 20cm. When the temperature inside the electrical cabinet rises, it must be ensured that the temperature difference inside and outside of the cabinet does not exceed 10°C.
- In order to ensure the internal air circulation, a fan can be installed in the electric cabinet.
- The display panel must be installed where the sprayed coolant is unable to reach.
- When designing the electrical cabinet, please be consider reducing external electrical interference as much as possible to prevent the interference transmitted to the CNC.

1.3.2 System grounding requirements

The following grounding system is provided for CNC machine tools.

- **Signal Ground**
The signal ground provides the reference voltage (0V) of the telecommunication system.
- **Frame Ground**
Frame grounding is used for safety, and frame unit shell, panel and interface cables among each unit must be connected together. The frame ground can also suppress internal and external noise.
- **System earthing**
The system ground is used to connect the frame ground of the equipment and the unit to the ground.

1.3.3 Methods to prevent interference

In the design of CNC, anti-interference measures such as shielding electromagnetic radiation in the space, absorbing Impulse current, and filtering out power supply clutter etc. have been adopted, which can prevent the influence of external interference sources on the CNC itself to a certain extent. In order to ensure the stable operation of CNC, it is necessary to take the following measures when installing and connecting the CNC:

- ① The CNC should be far away from the equipment that causes interference (such as frequency converter, AC contactor, electrostatic generator, high voltage generator and power line segmentation device, etc.);
- ② To supply power to the CNC through the isolation transformer, the machine tool where the CNC is installed must be grounded, and the CNC and the drive unit must be connected to an independent grounding wire from the grounding point;
- ③ Interference suppression: Parallel RC circuit at both ends of the AC coil (as shown in Fig. 1-6), and install the RC circuit as close as possible to the inductive load; parallel freewheeling diodes at both ends of the DC coil in anti-parallel (Fig. 1-7); Connect a surge absorber in parallel to the winding end of the

AC motor (as shown in Fig. 1-8);

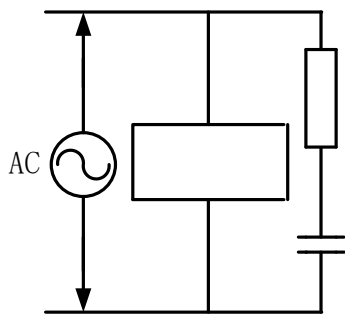


Fig. 1-6

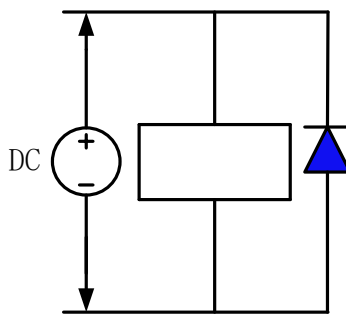


Fig. 1-7

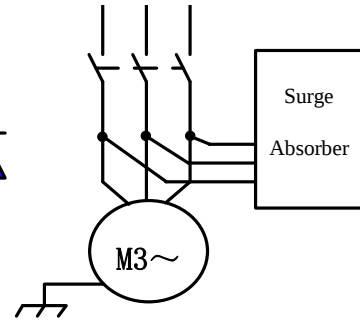


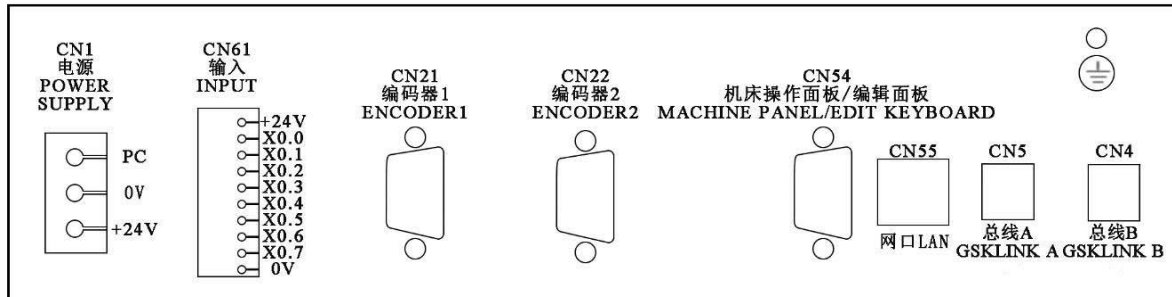
Fig. 1-8

- ④ The outgoing cable of CNC adopts twisted shielded cable or shielded cable. The shielding layer of the cable is grounded at the CNC side, and the signal wire should be as short as possible;
- ⑤ In order to reduce the mutual interference between CNC signal cables and strong current cables, the following principles should be followed when wiring.

Group	Cable Types	Wiring Requirements
A	AC power cords	Bundle the cables of group A with groups B and C, leaving a distance of at least 10cm between them; or electromagnetically shield the cables of group A.
	AC coils	
	AC contactor	
B	DC coil (24VDC)	Separately bundle the group B cables and group A cables; or shield the group B cables; the farther the group B cables and the group C cables are, the better.
	DC relay (24VDC)	
	Cable between CNC and strong electric cabinet	
	Cable between CNC and machine tool	
C	Cable between CNC and servo drive unit	Separate group C and group A cables and keep the distance between them at least 10cm; or shield group C cables; the distance between group C cables and group B cables is at least 10cm.
	Encoder feedback cable	
	Position feedback cable	
	Hand pulse cable	

1.4 CNC host

The interfaces of the system host are divided into 1 interface (USB interface) on the CNC front panel and 8 interfaces on the rear panel.



1.4.1 GSK-Link bus interface

The bus interfaces of the system host are CN5 and CN4 (GSK-LinkA and GSK-LinkB), which are connected to the feed servo drive unit, spindle drive unit and expansion I/O unit with GSK-Link bus communication function.

The GSK-Link bus communication connection line is shown as in Fig. 1-9.

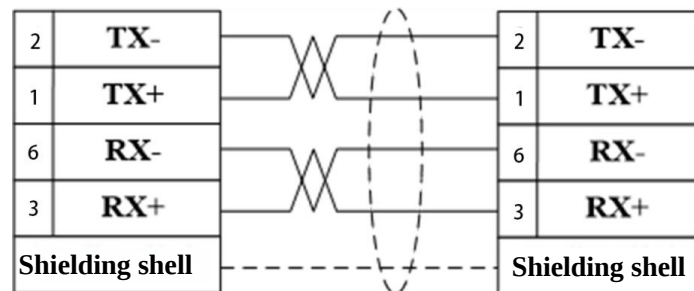


Fig. 1-9 GSK-Link communication connection

1.4.2 Spindle encoder interface

The system has two encoder input interfaces (CN21, CN22). The pinout diagram of the CN21 interface is shown in Figure 1-10, and the pinout diagram of the CN22 interface is shown in Figure 1-11. When the spindle encoder is not used but the GSK-Link spindle is used, use the GSK-Link interface to read the encoder speed.

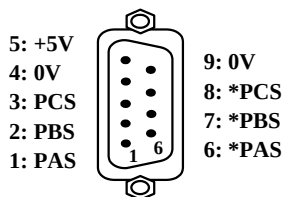


Fig. 1-10 CN21 encoder interface

(9-pin D-pin socket)

Signal Definition	Signal Description
* PAS/PAS	Encoder A phase pulse
* PBS/PBS	Encoder B phase pulse
* PCS/PCS	Encoder C phase pulse

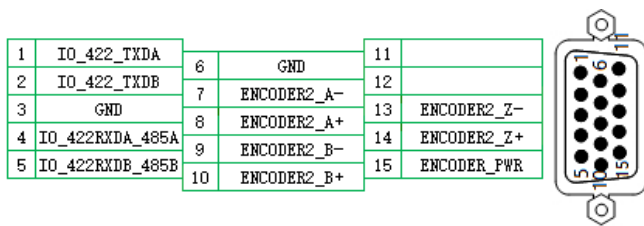


Fig. 1-11 CN22 encoder interface
(15-pin Female Socket)

Signal Definition	Signal Description
ENCODER2_A-	Encoder A phase pulse
ENCODER2_A+	
ENCODER2_B-	Encoder B phase pulse
ENCODER2_B+	
ENCODER2_Z-+	Encoder C phase pulse
ENCODER2_Z-+	

Signal description

*PCS/PCS, *PBS/PBS, *PAS/PAS are the differential input signals of the encoder's C-phase, B-phase, and A-phase respectively; *PAS/PAS, *PBS/PBS are quadrature square waves with a phase difference of 90°. The highest signal frequency is less than 1.2MHz; the line number of the encoder used by the system host is set via the system parameter NO.3720 (the line number of the spindle encoder).

Spindle encoder interface connection

The connection of the spindle encoder is shown in Fig. 1-12, and the twisted pair is used for the connection (take the Changchun Yiguang ZLF-12-102.4BM-C05D Encoder as an example).

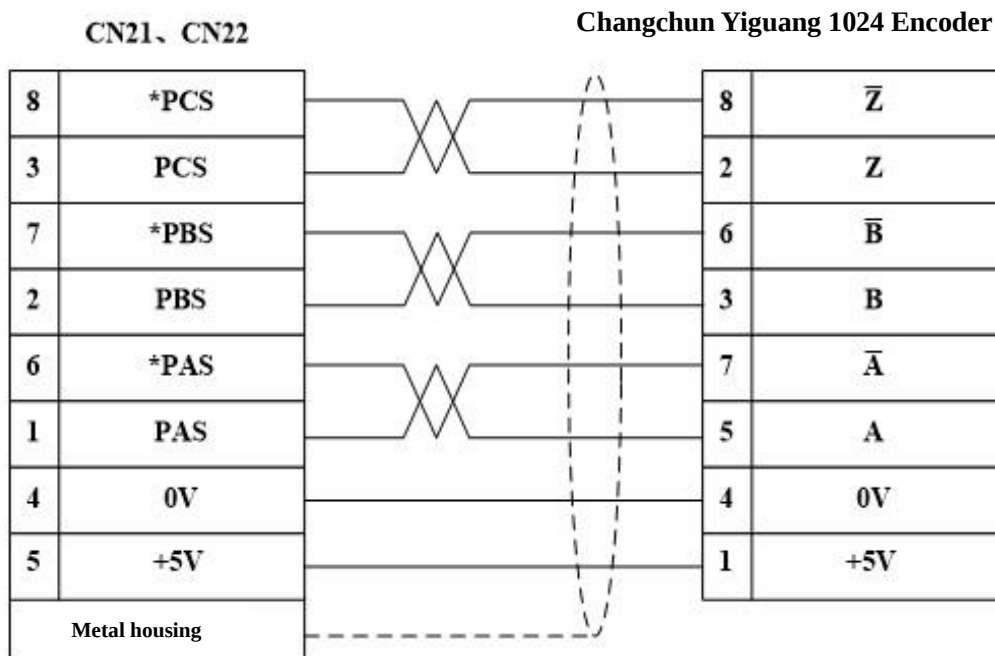
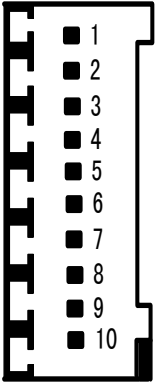


Fig. 1-12 Connection of the spindle encoder

1.4.3 High-speed input interface

The system host is equipped with one input signal high-speed I/O interface CN61, the address is X0.0~X0.7.

Table 1-1

Input Port CN61	CN61 Pin Number	PLC Address
 CN61	1	+24V
	2	X0.0
	3	X0.1
	4	X0.2
	5	X0.3
	6	X0.4
	7	X0.5
	8	X0.6
	9	X0.7
	10	GND

1.4.4 Communication interface

The system and the machine tool operation panel are connected by means of communication. The interface pins are shown in Fig. 1-13.

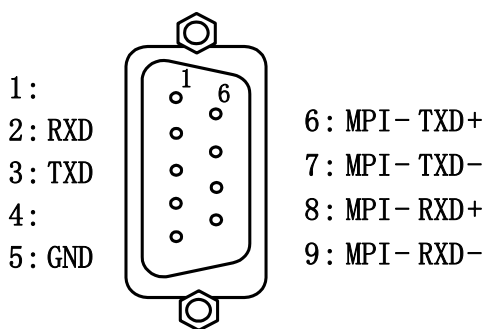


Fig. 1-13

Signal	Note
MPI-RXD+	Operation panel dedicated communication signal: accept data differential signal
MPI-RXD-	Operation panel dedicated communication signal: accept data differential signal
MPI-TXD+	Operation panel dedicated communication signal: send data differential signal
MPI-TXD-	Operation panel dedicated communication signal: send data differential signal
GND	Power 0V

The connection between the system host and the machine tool operation panel is shown in Fig. 1-14.

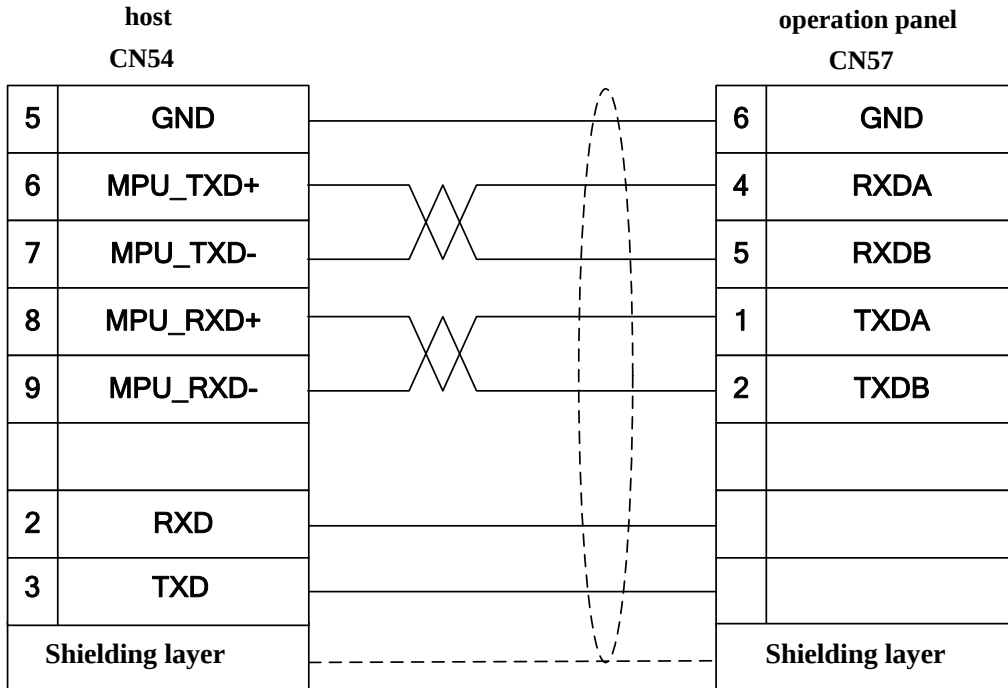


Fig. 1-14

Note:CN54 (DB9 double row hole), CN57(DB15 three row holes)

1.4.5 Power socket

The system host uses DC 24V power supply, and the power interface definition is shown in Fig. 1-15.

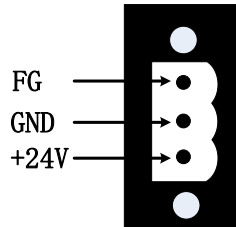


Fig. 1-15 power interface CN1 pin definition

1.4.6 Network port

The network interface (standard interface) is shown in Table 1-2.

Table 1-2

Pin Number	Signal	Pin Number	Code Signal
1	TXDLAN+	9	LINK_LED
2	TXDLAN-	11	LAN_LED
3	RXDLAN+	10, 12	VDD33
6	RXDLAN-	13, 14	Chassis land

Note: TXD+ and TXD- are differential signals, RXD+ and RXD- are differential signals, and both require twisted-pair connection.

1.4.7 Panel USB interface

The USB interface (standard interface) is shown in Table 1-3.

Table 1-3

Pin Number	Signal
1	VCC (+5V)
2	USB_DN0
3	USB_DP0
4	GND
5, 6	Chassis land

1.5 CNC operation panel

1.5.1 Band switch dedicated interface

The dedicated interface for the band switch is shown in Figure 1-16 (CN371~CN378), and the pin and address distribution is shown in Figure 1-17.

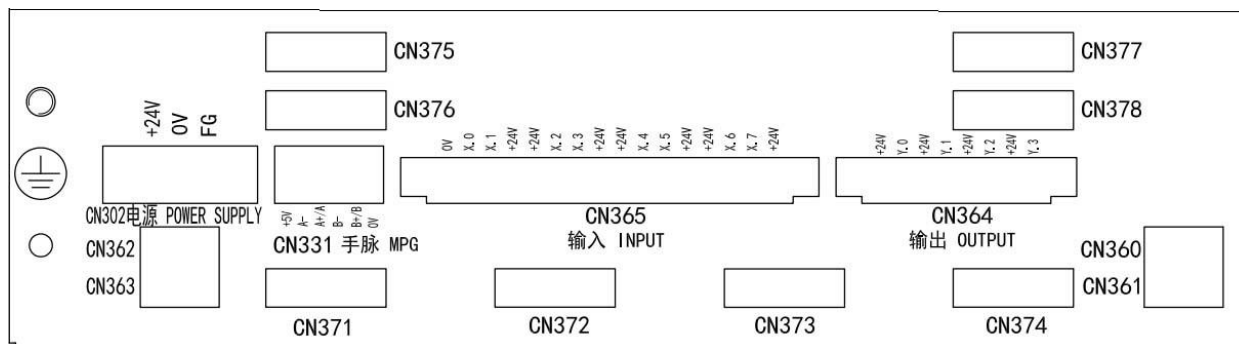


Fig. 1-16

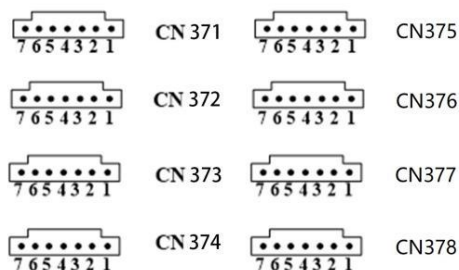


Fig. 1-17

Table 1-4

Interface	Pin Number	PLC Address	Function	Descriptions	Interface	Pin Number	PLC Address	Function	Descriptions
CN371	1	X30.0			CN373	1	X31.0	CN73_A	Spindle override band switch 0
	2	X30.2				2	X31.2	CN73_B	Spindle override band switch 1
	3					3	NC	Suspension	Suspension
	4	+5V				4	+5V	5V	Power supply 5V
	5	X30.3				5	X31.3	CN73_E	Spindle override band switch 3
	6	X30.1				6	X31.1	CN73_F	Spindle override band switch 2
	7	INH				7	INH	CN73_INH	Band switch inhibit bit
CN372	1	X30.4			CN374	1	X31.4	CN74_A	Feed rate override band switch 0
	2	X30.6				2	X31.6	CN74_B	Feed rate override band switch 1
	3					3	NC	Suspension	Suspension
	4	+5V				4	+5V	5V	Power supply 5V
	5	X30.7				5	X31.7	CN74_E	Feed rate band switch 3
	6	X30.5				6	X31.5	CN74_F	Feed rate band switch 2
	7	INH				7	INH	CN74_INH	Band switch inhibit bit
CN375	1	X32.4			CN377	1	X33.0	OFF	
	2	X32.6				2	X33.1	X	
	3					3	X33.2	Y	
	4	+5V	5V	Power Supply +5V		4	+5V	5V	
	5	X32.7				5	X33.3	Z	
	6	X32.5				6	X33.4	4	
	7	INH		Band Switch Inhibit Bit		7	X33.5	C	

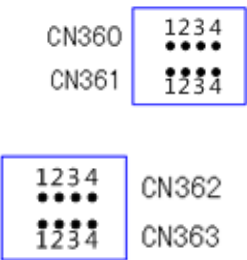
Interface	Pin Number	PLC Address	Function	Descriptions	Interface	Pin Number	PLC Address	Function	Descriptions
CN376	1	X32.0		Turret Band Switch 0	CN378	1	X34.0	X1	
	2	X32.2		Turret Band Switch 2		2	X34.1	X10	
	3					3	X34.2	X100	
	4	+5V	5V			4	+5V	5V	
	5	X32.3		Turret Band Switch 3		5	X34.3	X1000	
	6	X32.1		Turret Band Switch 1		6	X34.4		
	7	INH		Band Switch Inhibit Bit		7	X34.5		

Note: The dedicated interface for the band switch is a 5V input signal, which can only be connected to the band switch. Wrong connection may cause the system to burn out.

1.5.2 Dedicated interface for external key

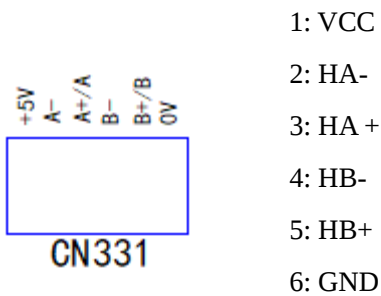
The pin and address distribution of the dedicated interface (CN360~363) for external keys on the operation panel is shown in Table 1-5.

Table 1-5

Interface	Sequence Number	Pin Number	PLC Address
	360	1	+24V
		2	+24V
		3	Y30.6
		4	X34.6
	361	1	+24V
		2	+24V
		3	Y30.7
		4	X34.7
	362	1	+24V
		2	+24V
		3	Y30.0
		4	X34.0
	363	1	+24V
		2	+24V
		3	Y30.1
		4	X34.1

1.5.3 MPG interface

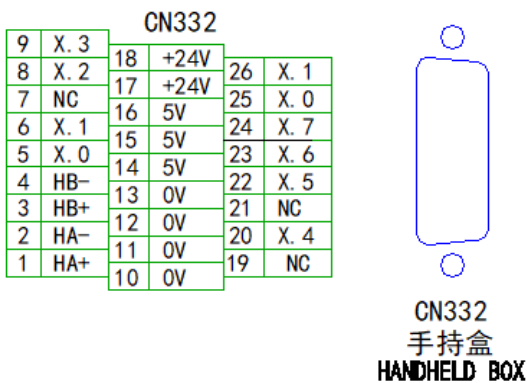
The definition of MPG interface CN331 is shown in Fig. 1-18.



Signal Definition	Signal Description
HA+, HA-	MPG A-phase Signal Input
HB+, HB-	MPG B-phase Signal Input
GND	0V
VCC	+5V

Fig. 1-18 CN331 MPG interface

The definition of the handheld box interface CN332 is shown in Fig. 1-19.



Signal Definition	Signal Description
HA+, HA-	MPG A-phase signal input
HB+, HB-	MPG B-phase signal input
X37.0 ~ X38.1	PLC signal address, switch input

Fig. 1-19 CN332 MPG interface (26-pin D-pin socket)

Signal Description

HA+, HA- and HB+, HB- are the MPG A-phase and B-phase differential input signals respectively.

The X37.0~X38.1 signals are the input addresses defined by the PLC interface, which are used for the axis selection and gear signal input of the external handheld box.

X37.0~X38.1 signals are valid for high level input.

The connection to the MPG interface is shown in Fig. 1-20 and Fig. 1-21.

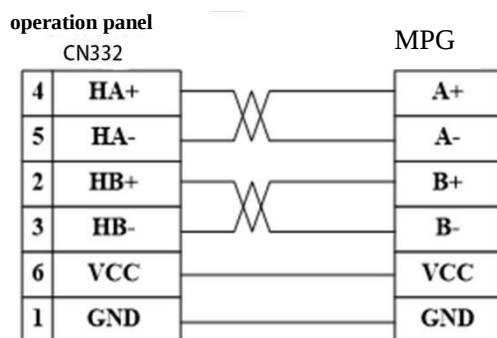


Fig. 1-20 Connection diagram of CN332 and MPG

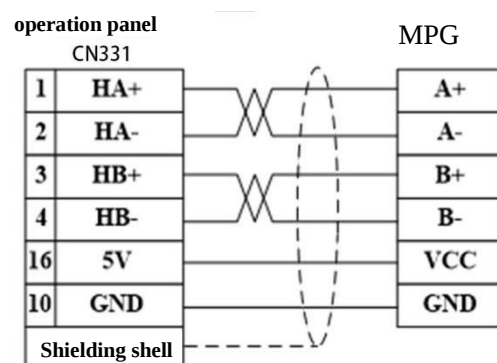


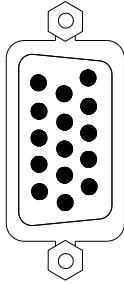
Fig. 1-21 Connection diagram of CN331 and MPG

Note: If a non-differential MPG is connected, leave the HA- HB- signal blank without connecting.

1.5.4 Communication interface

The system and the machine tool operation panel are connected by means of communication.

The definition of the communication interface of the machine operation panel is shown in Fig. 1-22.



Pin	Signal	IN/OUT	Descriptions
1	RXDA	IN	Accept data differential signal
2	RXDB	IN	Accept data differential signal
4	TXDA	OUT	Send data differential signal
5	TXDB	OUT	Send data differential signal

Fig. 1-22 Standard machine operation panel interface CN357
(15-pin D-pin socket)

Note: Refer to 1.4.4 of this chapter for the connection line with CNC.

1.5.5 General input/output address

The general input/output signals are located on the back cover of the machine operation panel and lead out to the terminal block. The addresses reserved for users are 8 input points X36 and 4 output points Y31.

1.5.6 Machine panel power interface

The operation panel of the machine tool adopts DC 24V power supply, and the power interface definition is shown in Fig. 1-23.

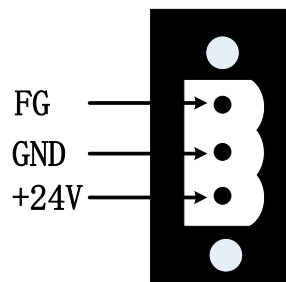


Fig. 1-23 Definition of CN302 pin of operation panel power interface

1.6 I / O unit

1.6.1 Bus interface CN5/CN5

Buses A and B of interfaces CN5/CN4 are the bus interfaces of the I/O unit.

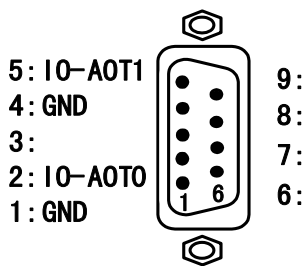
1.6.2 Communication interface CN52

485 communication interface, reserved interface.

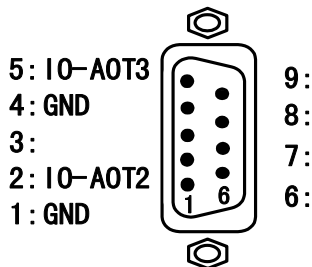
1.6.3 Spindle CN41 and spindle CN42

1.6.3.1 IOR-44T and IOR-44F

The IOR-44T and IOR-44F expansion I/O units contain 4 analog voltage output interfaces, which are distributed in the CN41 and CN42 interfaces. They are used to connect to non-bus type spindles that need to output analog voltage, while bus type spindles do not Connection, the interface is defined as follows.



CN41 interface



CN42 interface

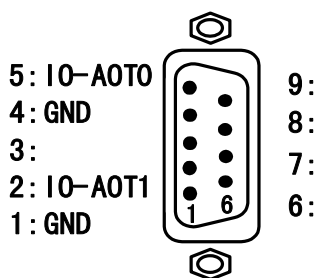
Fig. 1-24

Signal Definition	Signal Description
1: GND	The first analog voltage output ground
2: IO-AOT0	0V~+10V first analog voltage output
4: GND	The second analog voltage output ground
5: IO-AOT1	0V~+10V second analog voltage output

Signal Definition	Signal Description
1: GND	The third analog voltage output ground
2: IO-AOT2	0V~+10V the third analog voltage output
4: GND	The fourth analog voltage output ground
5: IO-AOT3	0V~+10V the fourth analog voltage output

1.6.3.2 IOR-21F

The IOR-21F expansion I/O unit contains 2 analog voltage output interfaces, which are distributed in the CN42 interface. They are used to connect non-bus type spindles that need to output analog voltage, while bus type spindles do not need to be connected. The interface is defined as follows.



CN42 interface

Fig. 1-25

Signal Definition	Signal Description
4: GND	The first analog voltage output ground
5: IO-AOT0	0V~+10V the first analog voltage output
1: GND	The second analog voltage output ground
2: IO-AOT1	0V~+10V the second analog voltage output

1.6.4 Power interface CN1

The expansion I/O unit uses DC 24V power supply, and the power interface definition is shown in Fig. 1-26.

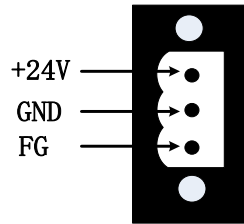
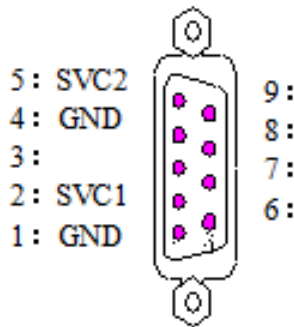


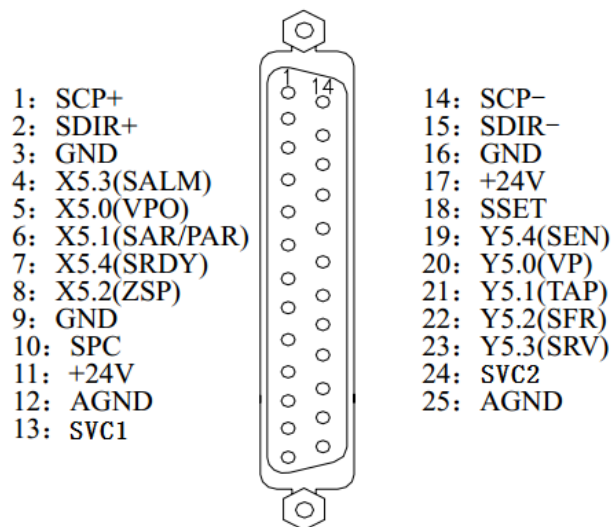
Fig. 1-26

1.6.5 Spindle Interface (Analog Output: 0~10V)



Pin	Signal	Signal Description
1	GND	The first analog voltage output ground
2	SVC1	the first analog voltage output
4	GND	The second analog voltage output ground
5	SVC2	the second analog voltage output

1.6.6 Pulse Axis Interface (IOR-20TP and IOR-23TP)



Pulse Axis Interface (DB25 Female Socket)

Signal Definition	Signal Description	The Function of Standard PLC Address Definition
SCP+, SCP-	Command Pulse Signal	/
SDIR+, SDIR-	Command Direction Signal	/
X5.3(SALM)	Drive Unit Alarm Signal	Drive Unit Alarm Signal
X5.4(SRDY)	Servo Ready Signal	Servo Ready Signal
SSET	Pulse Inhibit Signal	/
Y5.4(SEN)	Axis Enable Signal	Axis Enable Signal
SPC	Zero Point Signal	/
SVC1	0V~+10V Analog Voltage Output	/
AGND	Analog Output GND	/
X5.0(VPO)	PLC Signal Address, Digital Input	Spindle Speed/Position Status Signal
X5.1(SAR/PAR)	PLC Signal Address, Digital Input	Spindle Position/Speed Arrival Signal
X5.2(ZSP)	PLC Signal Address, Digital Input	Spindle Zero - Speed Output Signal
Y5.0(VP)	PLC Signal Address, Digital Output	Spindle Speed/Position Switching Signal
Y5.1(TAP)	PLC Signal Address, Digital Output	Spindle Speed Loop Second Gain Selection Signal for Tapping
Y5.2(SFR)	PLC Signal Address, Digital Output	Spindle Forward Rotation Signal
Y5.3(SRV)	PLC Signal Address, Digital Output	Spindle Reverse Rotation Signal
SVC2	0V~+10V Analog Voltage Output	/

1.7 Use of CNC input and output signals

The input and output signals of CNC include high-speed input and output signals on the CNC host, special signals for band switches on the machine operation panel, special signals for external keys on the operation panel, and general input and output signals on the operation panel.

1.7.1 Input signal

The input signal refers to the signal from the electrical circuit of the machine tool or the panel of the machine tool to the CNC. When the interface input point is connected to +24V (the band switch dedicated interface is connected to +5V), the input is valid, and the corresponding X address signal state is 1; When the interface input point is disconnected from +24V, the input is invalid, and the corresponding X address signal state is 0. The contact of the input signal on the machine side should meet the following conditions:

Contact capacity: DC30V, 16mA or more.

Leakage current between contacts when open circuit: 1mA or less.

The voltage drop between the contacts when the circuit is closed: 2V or less (current 8.5mA, including the

voltage drop of the cable).

There are two ways to externally input the input signal: one uses a contact switch input, the connection is shown in Fig. 1-27.

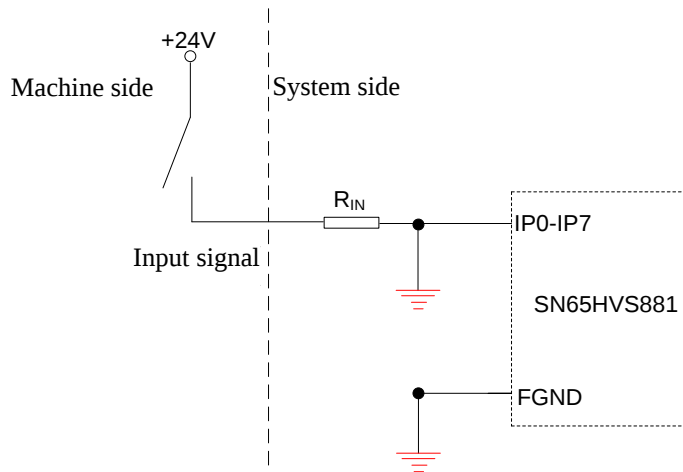


Fig. 1-27 Contact switch input

The other uses a non-contact switch (transistor) input, and the connection is shown in Fig. 1-28 and Fig. 1-29.

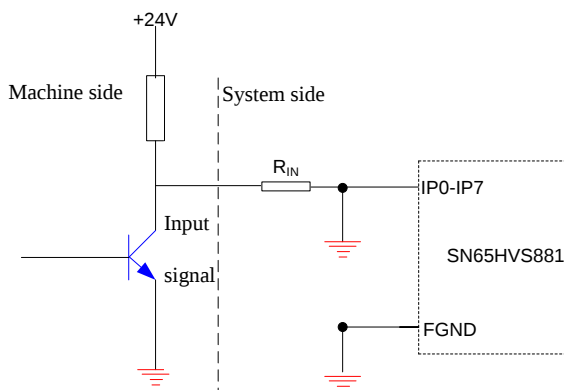


Fig. 1-28 NPN type connection

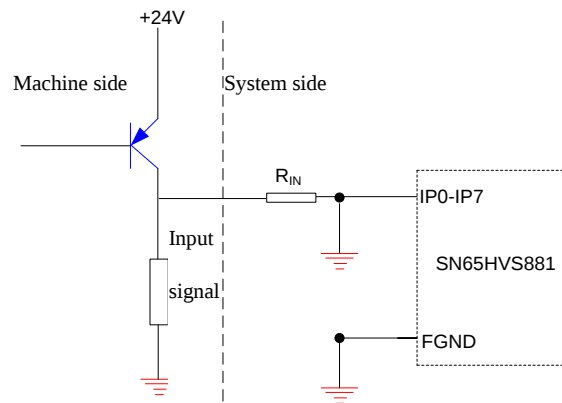


Fig. 1-29 PNP type connection

1.7.2 Output signal

The output signal is used to drive the relays and indicators on the machine electrical circuit side or the machine panel side. When the output is valid, the corresponding Y address output state is 1, and the output interface potential is 0V; when the output is invalid, the corresponding Y address output state is 0, the output interface is in a high impedance state. The circuit is shown in Fig. 1-30.

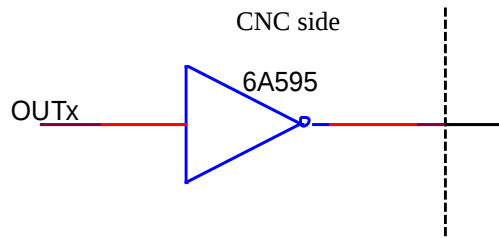


Fig. 1-30 Internal circuit structure diagram of the output signal

So the output signal has two output states: 0V output or high impedance. Typical applications are as follows.

➤ **Drive LED**

The output drive LED requires a resistor in series to limit the current flowing through the LED (usually about 10mA). As shown in Fig. 1-31.

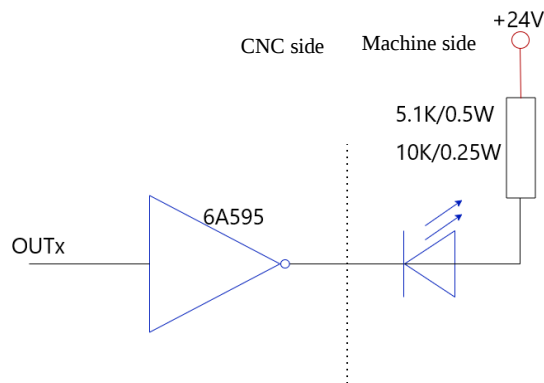


Fig. 1-31

➤ **Drive filament type indicator light**

The output drive filament type indicator light requires an external preheating resistor to reduce the current impact when it is turned on. The resistance of the preheating resistor is so large that the indicator light does not light up, as shown in Fig. 1-32.

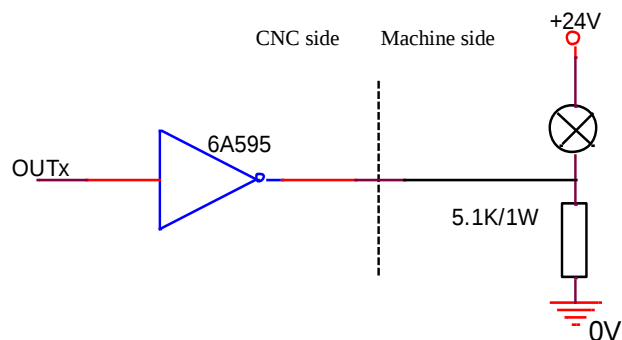


Fig. 1-32

➤ **Drive inductive loads (such as relays)**

The output drives an inductive load. At this time, a freewheeling diode needs to be connected near the coil to protect the output circuit and reduce interference. As shown in Fig. 1-33.

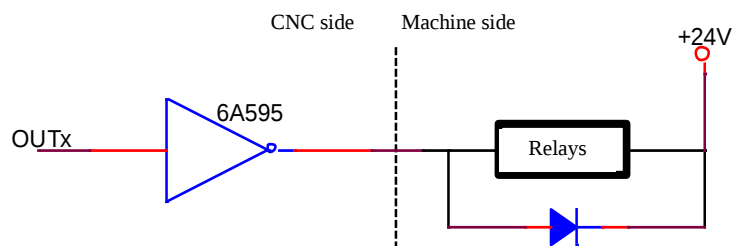


Fig.1-33

1.8 Use of I/O unit input and output signal

The configuration of different models of I/O unit is different, the configuration of each model is shown in Table 1-6.

Table 1-6

Model	Input Signal Point	Output Signal Point	Analog Amount Input	Analog Amount Output	Input Level	Output Level
IOR-04T	48 points	32 points	0	0	High	Low
IOR-44T	48 points	32 points	0	4 way	High	Low
IOR-21F	24 points	16-point	0	2 way	High	High
IOR-44F	48 points	32 points	0	4 way	High	High
IOR-49F8	72 points	48 points	0	4 way	High	High
IOR-26T	48 points	48 points	0	2 way	High	Low
IOR-20TP	16 points	8 points	0	2 way	High	Low
IOR-23TP	40 points	24 points	0	2 way	High	Low

1.8.1 Input signal

Please refer to 1.6.1 of this chapter for how to use it.

1.8.2 Output signal

1.8.2.1 Output signal is low level

Please refer to 1.6.2 of this chapter for how to use it.

1.8.2.2 Output signal is high

The output signal is used to drive the relays and indicators on the machine electrical circuit side or the machine panel side. When the output is valid, the corresponding Y address output state is 1, and the output interface potential is +24V; when the output is invalid, the corresponding Y address output state is 0, the output interface behaves in a high impedance state. The circuit is shown in Fig. 1-34.

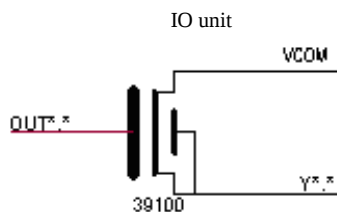


Fig. 1-34 The internal circuit structure diagram of the output signal

Therefore, the output signal has two output states: +24V output or high impedance. Typical applications are as follows.

➤ Drive LED

The output drive LED requires a resistor in series to limit the current flowing through the LED (usually about 10mA). As shown in Fig. 1-35.

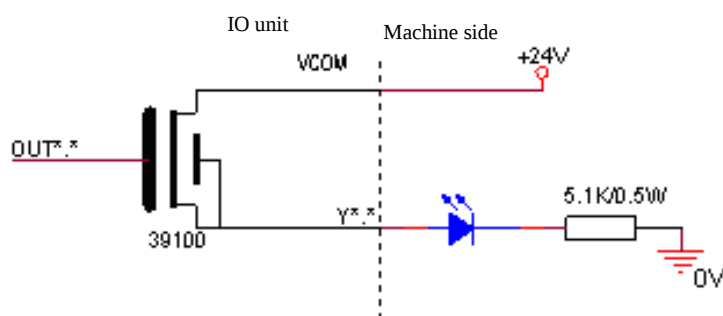


Fig. 1-35

➤ Drive filament type indicator light

The output drive filament type indicator light requires an external preheating resistor to reduce the current impact when it is turned on. The preheating resistor has a resistance value so that the indicator light does not light up as a principle, as shown in Fig.1 -36.

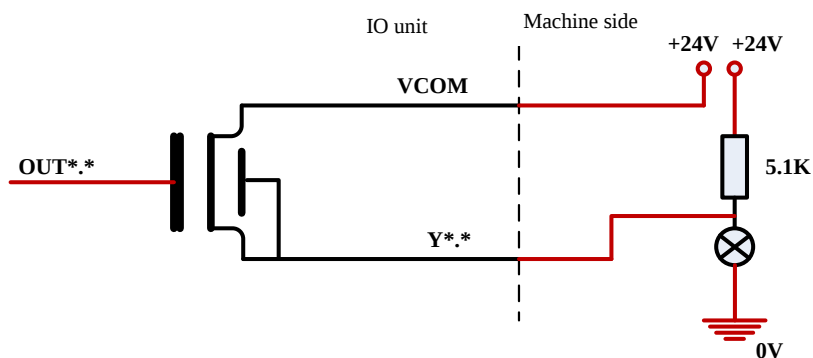


Fig. 1-36

➤ Drive inductive loads (such as relays)

The output drives an inductive load. At this time, a freewheeling diode needs to be connected near the coil to protect the output circuit and reduce interference.

As shown in Fig. 1-37.

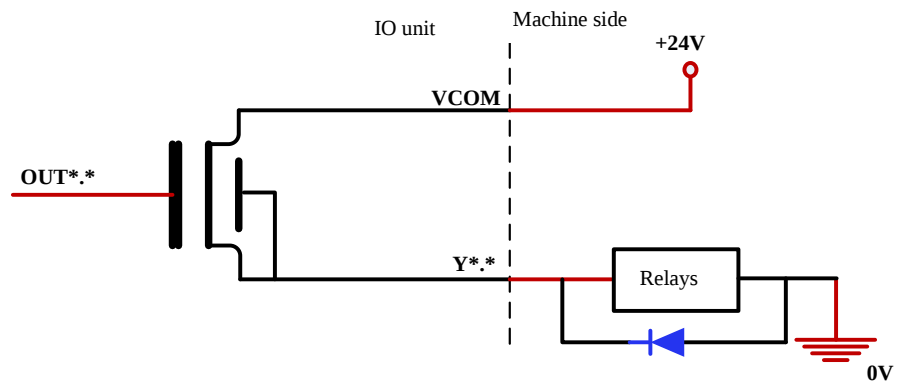


Fig. 1-37

Volume 2 Debugging

Chapter 1 Machine Tool Debugging and Operation

1.1 Parameter setting

The modification, backup and restoration of the system parameters and servo parameters must be at the equipment management level (level 3) or higher, with the parameter switch turned on, and can only be carried out in the input mode. The operation to turn on the parameter switch is shown in Figure 1-1..

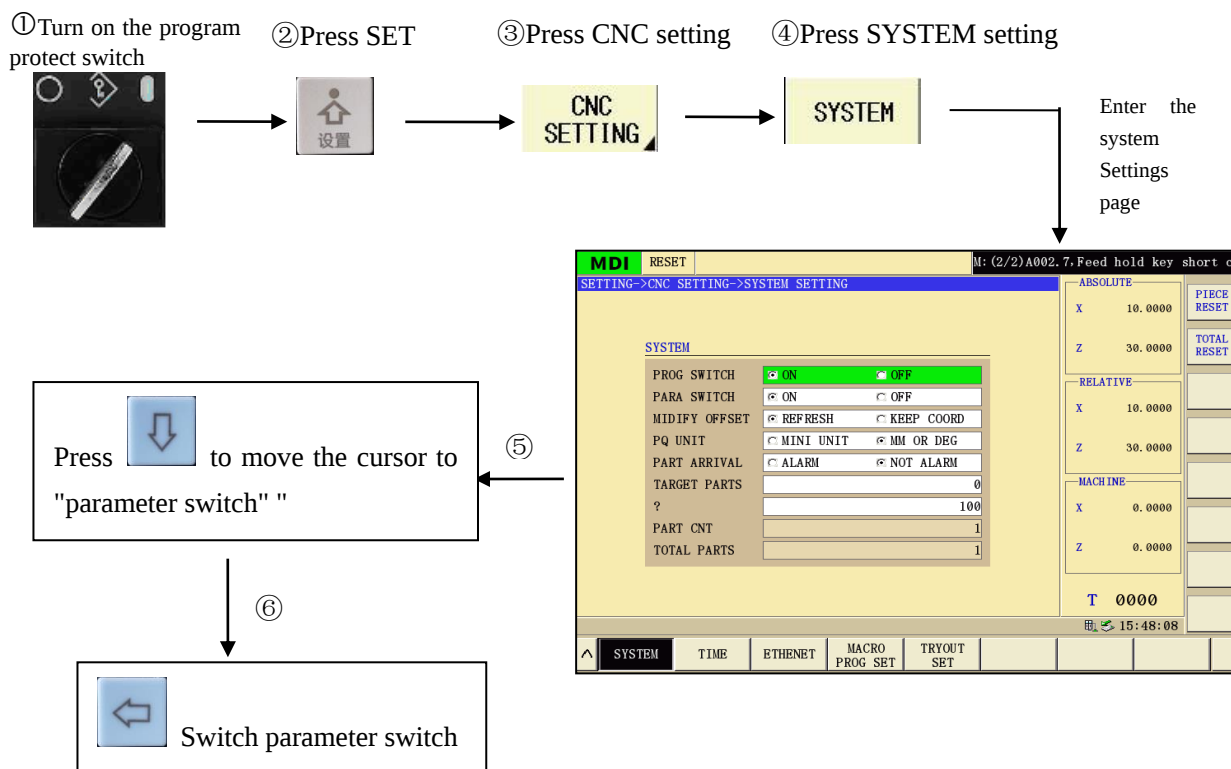


Fig. 1-1

Note 1: After the system parameters are modified, some parameters can take effect immediately, and some parameters can only take effect after the system is powered on again. For details, see "Appendix 1 Parameter Description".

Note 2: To view and modify the servo parameters in CNC, the correct connection of the servo system and the correct configuration of the servo slave number should be ensured.

1.1.1 System parameters

Press in sequence: to enter the system parameter setting interface.

It can set system parameters on this page. It can back up the parameters currently set by the user under the secondary authority, restore the parameters to the system default parameters or restore the parameters backed up

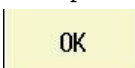
by the user.

Note: Before the machine is debugged, you can select and call the corresponding parameters according to the configuration of the machine on the configuration page of the system parameters.

Parameter search:

Method 1: Use , ,  and  keys to select the parameter that needs to be viewed or modified.

Method 2: Search by parameter category, press the  soft key to select a different category name, and then press the  soft key to locate the cursor to the first parameter of this type.

Method 3: Search by parameter number, press  the soft key, input the parameter number to be selected, and then press  the soft key to locate the cursor on the parameter.

Bit parameter setting

Method 1:

① After finding and locating the parameter to be modified, press the  key to make the selected parameter in a modifiable state.

② Press the numeric key to input the 8-digit binary value to be modified, and then press the  key to confirm the completion of the setting (when the input value is less than 8 digits, the high digit is filled with 0).

③ Use , ,  and  keys to select other parameters to be set for setting.

Method 2:

① After finding and locating the parameter to be modified, use  and  keys to select the parameter bit to be modified.

② Press  the key repeatedly to switch the parameter bit between 0 and 1, and modify the value of the parameter bit.

③ Move the cursor to complete the setting.

④ Use , ,  and  keys to select other parameters to be set for setting.

Numeric Parameters Setting

① After finding and positioning the parameter to be modified, press the  key to make the selected parameter in a modifiable state.

② Press the numeric key to input the value to be set, and then press the  key to confirm the setting.

③ Use , ,  and  keys to select other parameters to be set for setting.

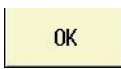
Parameter backup and recovery

Before modifying the parameters, the user can press the  soft key to back up the parameters.

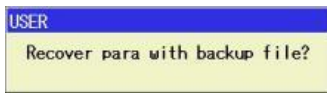
When the modified parameters are wrong or do not need to modify the parameters, press  the soft key to restore the parameters to the parameters backed up by the user before modification or to the system default parameters. Backup and restore can only be operated under Level 2 authority. After the parameters are restored, the power needs to be re-powered.

Backup of parameters:

① Press the  soft key  under the parameter page to display, and then enter the saved file name.

② Press to  backup the parameters currently set by the current user.

Recovery of parameters:

① Press  the soft key to display .

② Press  to restore the parameters previously saved by the user; press to  exit the restore parameter interface.

1.1.2 Servo parameters

Modification and saving of servo parameters

After the GSK-Link communication is normal, under the system page set, press   →  soft key to enter the servo parameter interface.

Fig. 1-2

Switching of Axis Parameter Page: Press , , to  switch between the X-axis, Z-axis, and Cs axis to display the servo parameters of the corresponding axis.

Modification of Parameter: Move the cursor to the position of the parameter that needs to be modified, and after inputting the parameter value, press the  key to complete the modification.

Save the Parameters: After modifying the servo parameters, press  to write the parameters into the servo. The parameter values that have been successfully modified will remain unchanged after the servo is powered on again.

Parameter Backup: Press to directly save the servo parameter backup to the save area.

Parameter Recovery: Press  to directly restore the parameters backed up to the save area to the currently used parameters.

Restore motor default parameters

- ① Press the  soft key on the servo configuration page.
- ② In the pop-up dialog box, select the currently used motor model, press  the soft key, and the system will call the default parameters of the corresponding motor and overwrite the currently saved

parameters.

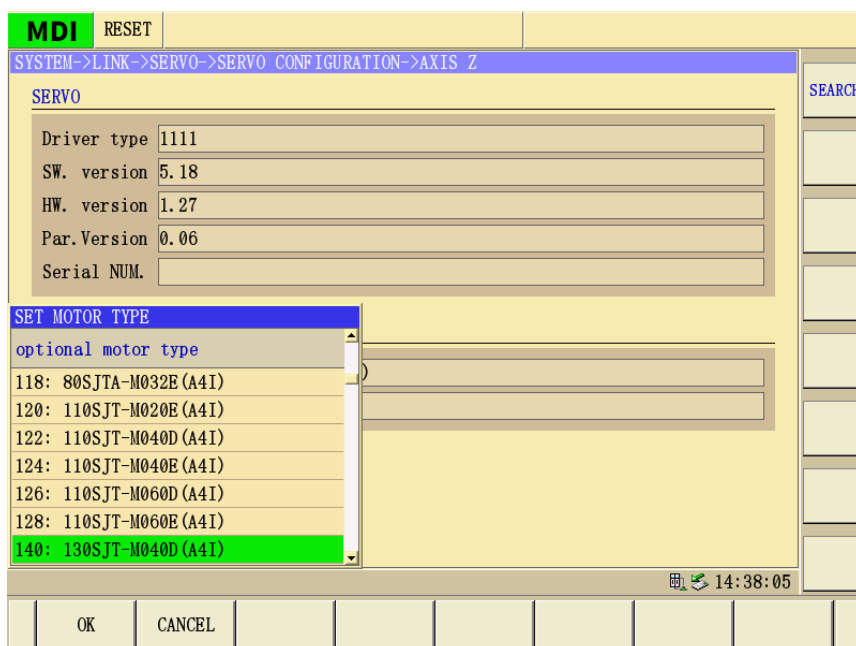


Fig. 1-3

1.1.3 I/O unit parameters

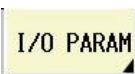
I/O unit configuration

After the GSK-Link communication bus is connected normally, if it is the first time to connect the I/O unit, since the parameters of the I/O unit in the system are all 0 by default, the system will alarm that the Ethernet communication is not established.

To manually conFig. the station address of the I/O unit in the system, first modify the parameter No.3050, set the number of I/O units that need to be connected, and then modify the parameters No.3051~No.3054, modify the station address to be conFig.d within the parameter range, disconnect Power restart,

press  → , modify the slave station number to be the same as No.3051~No.3054, and restart the I/O unit and CNC.

I/O unit parameter setting

After the GSK-Link communication is normal, under the system page set, press  →  →  soft key to enter the I/O unit parameter interface. If the system is connected to the I/O unit for the first time and the I/O unit parameters have not been set, the default PLC address is If it is empty, press  the button at this time, and the system will automatically assign the contacts to DI and DO starting from X100 and Y100, as shown in Fig. 1-4.

Setting of input contacts		Setting of output contacts	
CONTACT	PLC ADDRESS	CONTACT	PLC ADDRESS
DI01	X0100.0	DO01	Y0100.0
DI02	X0100.1	DO02	Y0100.1
DI03	X0100.2	DO03	Y0100.2
DI04	X0100.3	DO04	Y0100.3
DI05	X0100.4	DO05	Y0100.4
DI06	X0100.5	DO06	Y0100.5
DI07	X0100.6	DO07	Y0100.6
DI08	X0100.7	DO08	Y0100.7
DI09	X0101.0	DO09	Y0101.0
DI10	X0101.1	DO10	Y0101.1
DI11	X0101.2	DO11	Y0101.2
DI12	X0101.3	DO12	Y0101.3
DI13	X0101.4	DO13	Y0101.4
DI14	X0101.5	DO14	Y0101.5

Fig 1-4

● Input and output address settings

If you need to manually configure the address manually, move the cursor to the blank place on the right of the I/O type of DI, press **MODIFY**, and then enter 100, as shown in Fig. 1-5, press OK, the steps for other parameters are the same, just move the cursor to the desired setting Repeat the previous steps wherever you want. **Port type of output address is DO parameter, and setting method is the same as that of DI.**

Please input PLC address.

X 100

OK CANCEL

Please input PLC address.

Y

OK CANCEL

Fig. 1-5

Fig. 1-6

Note: Because the standard ladder diagram of the system has defined the I/O unit interface address, if you want to use the standard ladder diagram that comes with the system, you must config. the I/O unit according to the address defined by the ladder diagram. The following table is the I/O definition of the standard ladder diagram.

Table 1-1

I/O Unit Interface	Address Definition
CN66 input 1	X100.0~X100.7
CN67 input 2	X101.0~X101.7
CN68 input 3	X102.0~X102.7
CN71 input 4	X103.0~X103.7
CN72 input 5	X104.0~X104.7
CN73 input 6	X105.0~X105.7
CN69 output 1	Y100.0~Y100.7
CN70 output 2	Y101.0~Y107.7
CN74 output 3	Y102.0~Y102.7
CN75 output 4	Y103.0~Y103.7

1.2 U-disk operation

The U disk function of the system supports the two-way transmission of processing programs, PLC programs, parameters, tool compensation and screw compensation files. U disk can be operated under three pages of file management, program and ladder diagram.

1.2.1 File management page

When there is a U disk in the USB port of the system, on the  page, press the  soft key to enter the file management page, as shown in Fig. 1-7.

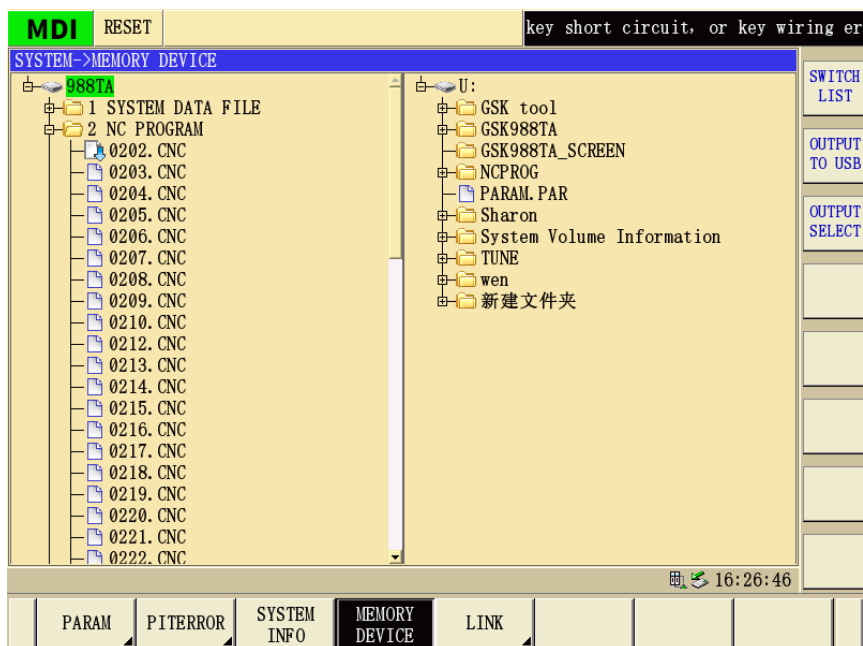


Fig. 1-7

This page can carry out two-way transmission of system files (system parameters, tool compensation, screw compensation, etc.), ladder files and part programs (CNC->U disk, U disk->CNC). The specific operations are as follows:

- ① Press  to switch the cursor back and forth between the system directory and the U disk directory.
- ② Use  the  cursor keys to select the file or directory to be copied, and press  the key to select (press the key repeatedly to select/cancel the switch).
- ③ Press the  or  soft key to copy the selected program to the local directory or U disk directory.
- ④ Click the  soft key to select the format for outputting the part program to the USB flash drive.

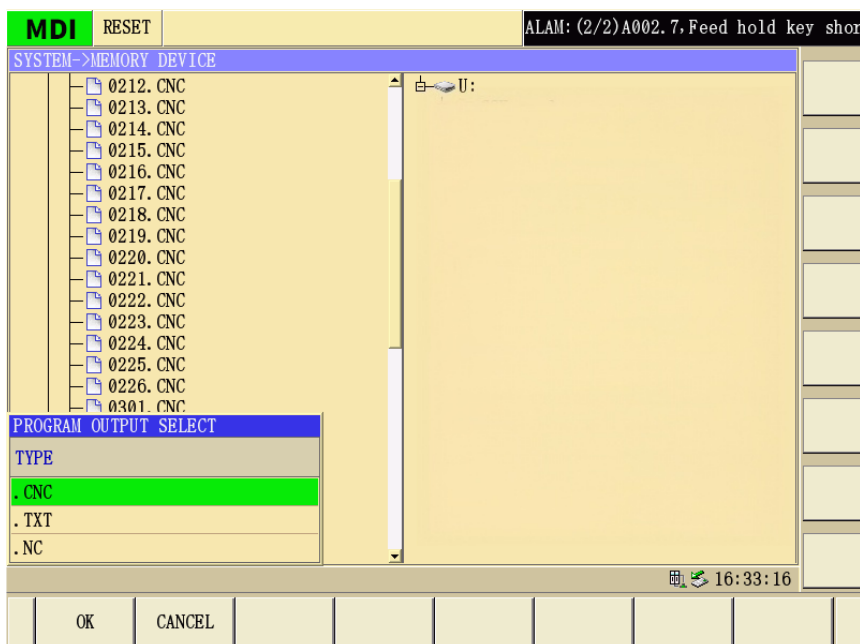


Fig. 1-8

1.2.2 Program page

When there is a U disk in the system USB port, press  to enter the program directory. Under the  page, you can load, open, copy, paste, create, save as, delete, rename, search and other operations on the programs in the U disk directory.

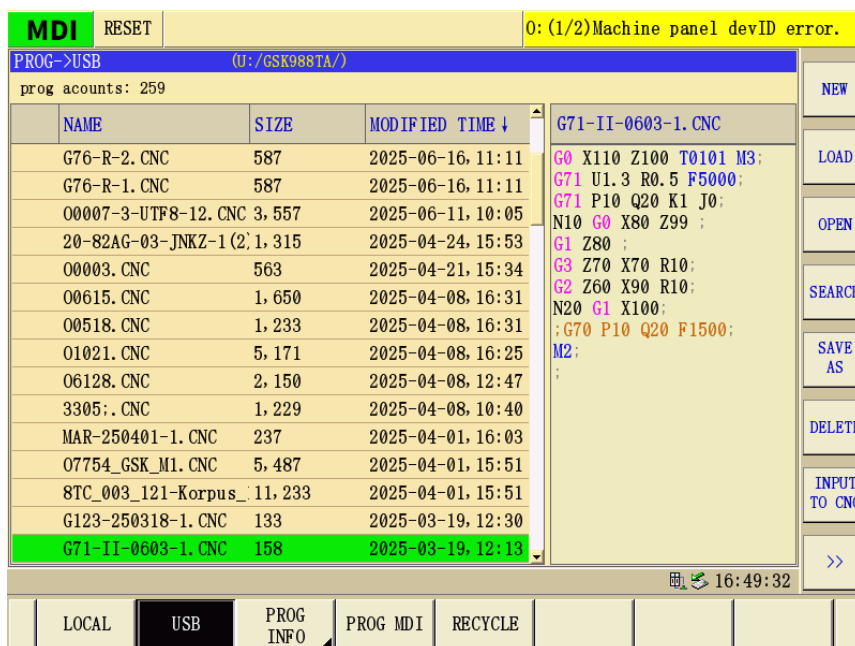


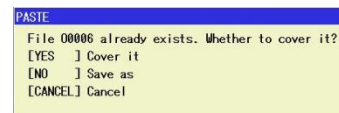
Fig. 1-9

Two-way transmission of programs

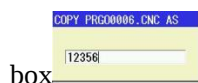
- ① Press **LOCAL** and **USB** to switch between system directory and U disk directory.

- ② Use the  and  keys to move the cursor to the program to be copied, and press the **OUTPUT TO USB** key.

or **INPUT TO CNC** soft key to copy the selected program to the U disk directory or local directory.



- ③ When the copied program already exists, a pop-up dialog box prompts (as shown on the right). Press the "Yes" soft key will overwrite the existing program; the "No" will pop up the copy file dialog



box, prompting to enter the program name and save as; press "Cancel" to cancel the operation.

Note 1: When transferring programs from U disk directory to local directory on the program page, the processing program must be placed in the "GSK25Ti" folder under the root directory of the U disk. The names of the folders created by different CNCs are different and can only be stored in the U disk directory. Read the processing program.

Note 2: When transferring processing programs from local directory to U disk directory, if there is no folder "GSK25Ti" in the U disk, a folder named "GSK25Ti" will be created automatically and the processing program will be output to this folder.

1.2.3 Ladder diagram page

When there is a U disk in the USB port of the system, press on the ladder diagram page



to switch to the U disk directory interface on the displayed interface.

You can copy the programs in the U disk directory to the local directory through the



soft keys, or

you can press



to copy the programs in the local directory to the U disk directory.

Take the U disk directory operation as an example, the specific steps are as follows:

- ① Press the  soft key to enter the U disk file directory;
- ② Select the ladder diagram program to be copied with  and  cursor keys, and press  the soft key to copy the selected program to the local directory.

Note 1: When transferring from U disk directory → local directory PLC on the ladder diagram page, the PLC program must be placed in the "GSK25Ti" folder under the root directory of the U disk (different types of CNC create folders have different names). The directory can read the PLC program.

Note 2: When performing PLC transfer from local directory → U disk directory, if there is no "GSK25Ti" folder in the U disk, a folder named "GSK25Ti" will be created automatically and the PLC program will be output to this folder.

1.3 PLC debugging operation



Press the  function key to enter the ladder diagram page set. This page mainly includes sub-pages such as version information, monitoring, PLC data, PLC status, program catalog, etc. You can view the content displayed under each page by pressing the corresponding soft key.

The content displayed when entering the ladder diagram page set is shown



in Fig. 1-10.



The page displays the version information of the ladder diagram, the currently running ladder diagram program and its running status, etc.



Fig. 1-10

1.3.1 PLC run and stop

Under the ladder diagram page set, press **PROGRAM** → **LOCAL** in sequence, and the interface that appears is shown in Fig. 1-11.

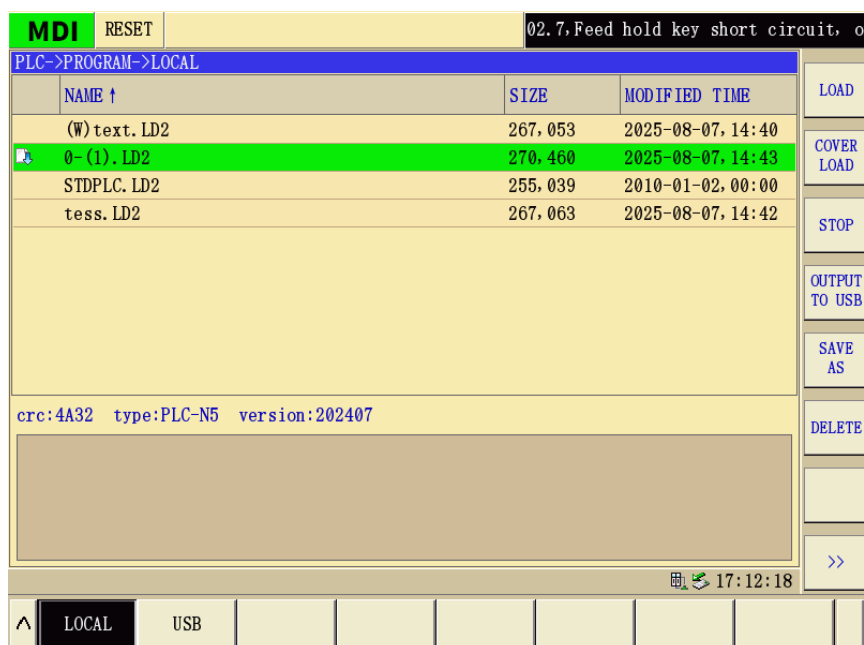


Fig. 1-11

On this page, you can use  the and  keys to select the PLC program, and then use the soft key

to open, execute, save, delete, stop and output the program to the U disk. Press  the soft key to create, and edit the program.

PLC program execution

Use  and  keys to select the PLC program, and then press  to run the selected PLC program.

Note: There is a mark in front of the PLC program currently running .

Stop of PLC program

Use the  and  keys to move the cursor to the program being executed, and then press , the system will be in a non-PLC running state, and the interface is shown in Fig. 1-12.

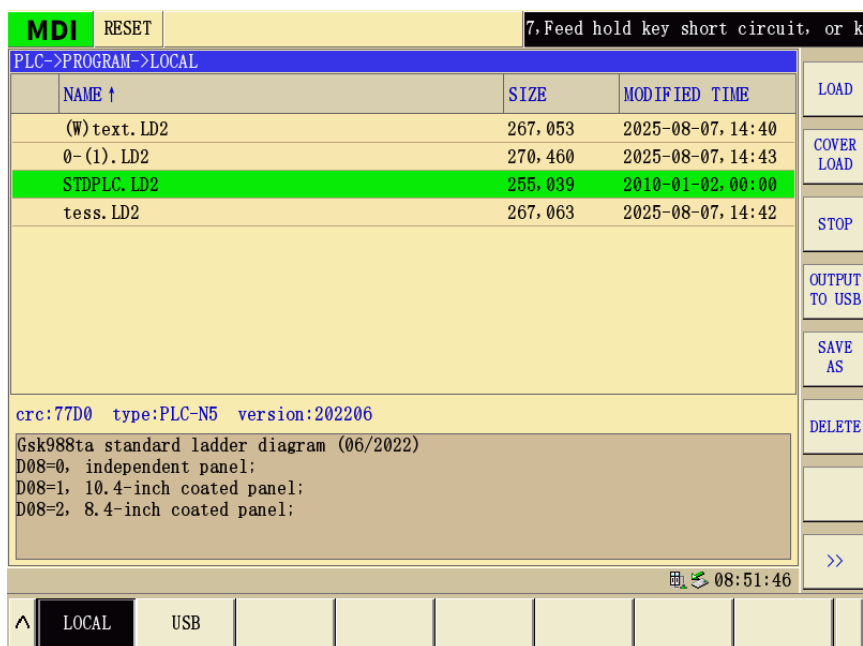


Fig. 1-12

1.3.2 PLC monitoring and diagnosis

PLC program status monitoring

In the ladder diagram page set, press the  soft key to enter the running monitoring display page of the currently running ladder diagram program, as shown in Fig. 1-13.

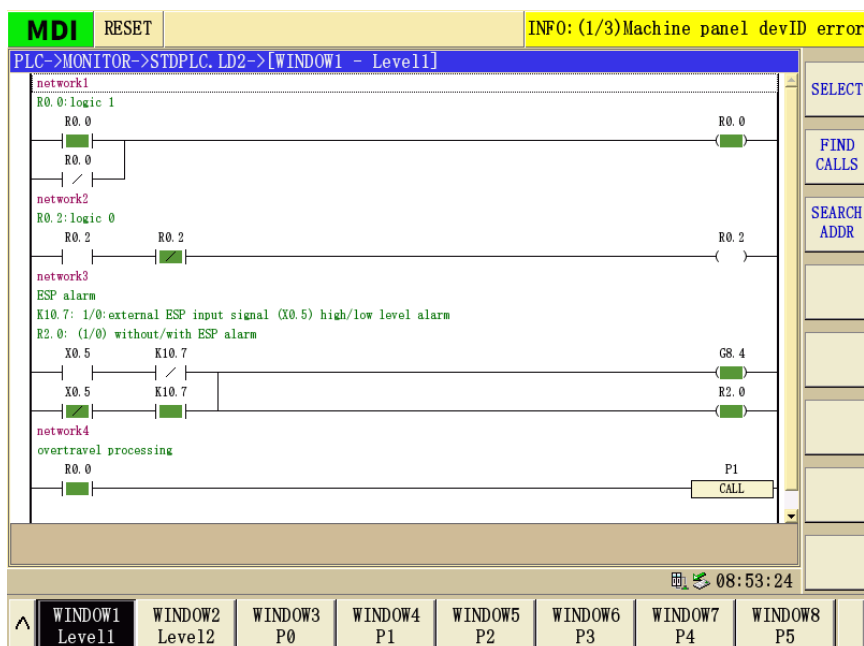
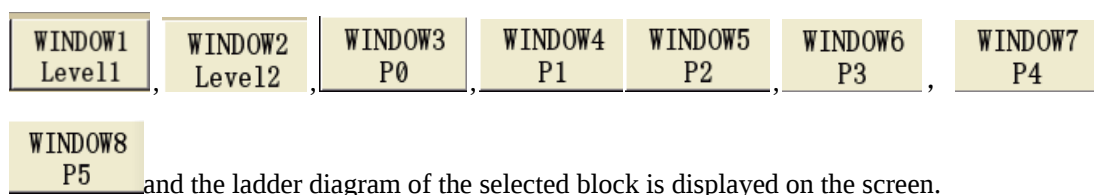


Fig. 1-13

On the monitoring page, you can view the on/off status of the current contacts and coils, the comments of the current contacts and coils, and the current values of timers and counters. When the contacts and coils are conducting, the background color is displayed in green, and the background color is the same as the background color of the window when not conducting. For example, it $\text{X0.5} \rightarrow \text{[green box]}$ means that the contact X0.5 is turned on, and $\text{Y25.2} \rightarrow \text{[white box]}$ that the coil Y25.2 is not turned on.

● Viewing window programs

Under the monitoring page, there are eight block windows for fast switching and monitoring:



and the ladder diagram of the selected block is displayed on the screen.

Note 1: The shortcut keys of window 1 to window 8 can quickly view the program block corresponding to the window.

Note 2: The program blocks corresponding to window 1 to window 8 can be changed as needed, but they will not be maintained after power-on. The first 8 program blocks in the ladder program are corresponding to the power-on default.

● Selection of window program block

- ① Select the window as needed.

- ② Press the  soft key, and the page shown in Fig. 1-14 is displayed at this time.

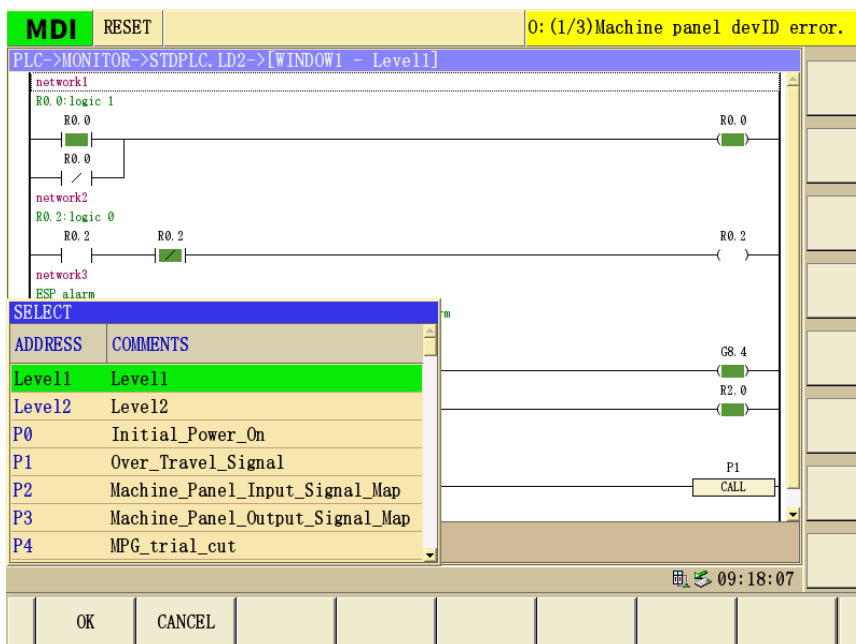
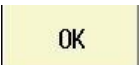


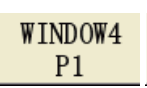
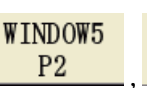
Fig. 1-14

- ③ Press , ,  and  keys to select the corresponding ladder block in the window.

- ④ Press the  soft key to confirm the selection and return to the previous menu, press the  soft key to cancel the selection operation and return to the previous menu.

● Search address and view call

- ① Select the program block window that needs to find the address and view the calling, that is,

press , , , , , ,  and  soft keys to select the window, let the corresponding program block ladder program be displayed in the window, and then search for the address and call.

- ② Press the  soft key to enter the search page, as shown in Fig. 1-15, and enter the address to be searched in the dialog box.

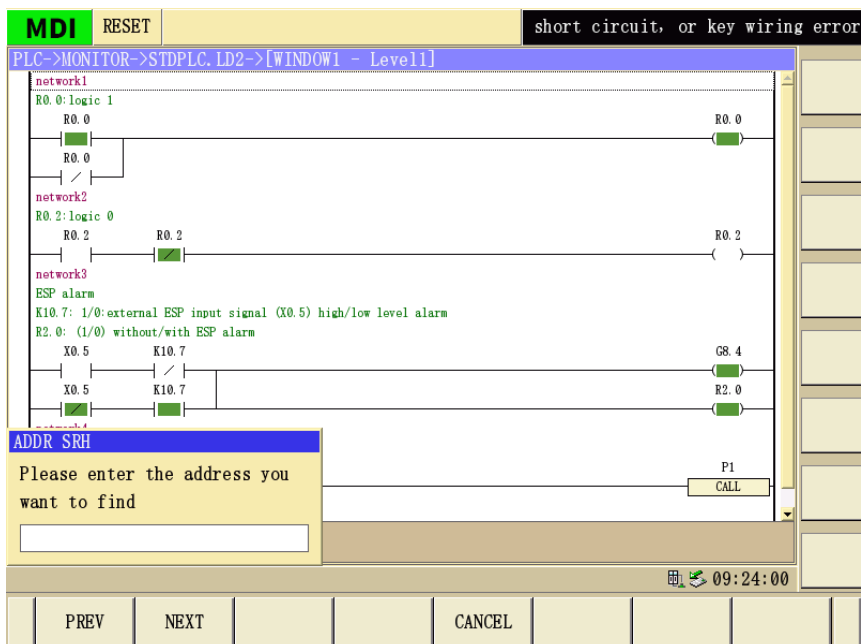


Fig. 1-15

- ③ Press **PREV** and **NEXT** soft keys respectively, and the cursor will jump to the previous or next page position where the address is found based on the current page position.
- ④ Press **FIND CALLS** this will pop up the program list that called the current subprogram. Use , the ,  and  keys to select the calling program. Press the **JUMP TO TUNE** button, and the cursor will jump to the position where the subprogram is called.

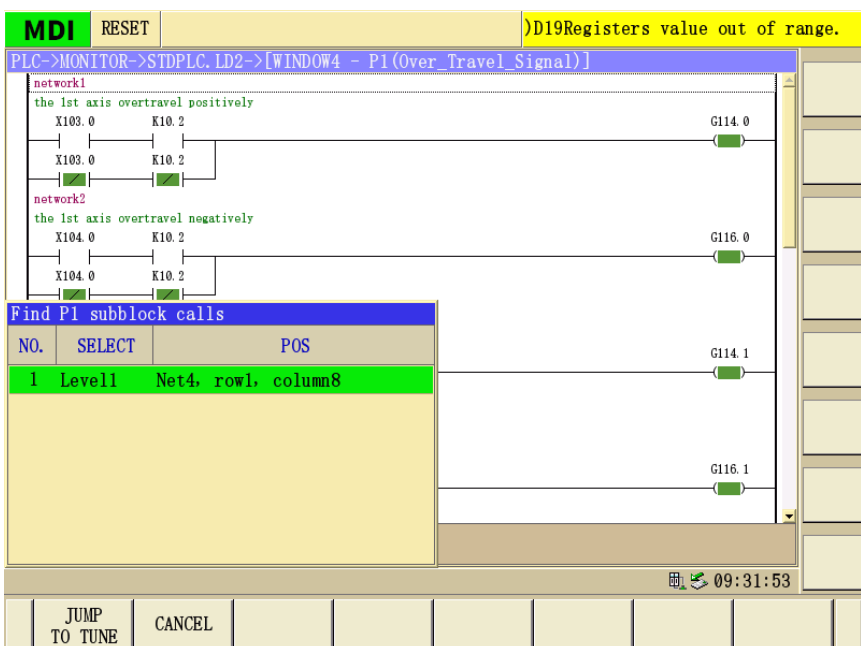


Fig. 1-16

PLC status diagnosis

PLC STATE

Under the ladder diagram page set, press the **PLC STATE** soft key to enter the PLC status display page. The display page is shown in Fig. 1-17.

The screenshot shows the 'PLC STATE' window with a title bar 'MDI RESET' and a status bar 'short circuit, or key wiring error'. The main area displays three columns of signal status: X (X000-X012), Y (Y000-Y012), and R (R000-R012). Each column has a 'DDRES' and 'VALUE' header. The 'X' column shows values like 00000000, 00000000, etc. The 'Y' column shows values like 00000000, 00000000, etc. The 'R' column shows values like 00100011, 00000001, etc. At the bottom, there is a comment line: 'X000 G37 G36 ESP G31 *** TOOL *** ***'. The bottom status bar shows 'X.Y.R', 'F.G.A', 'C.T', and a timestamp '09:44:09'.

Fig. 1-17

The screenshot shows the 'PLC STATE' window with a title bar 'MDI RESET' and a status bar 'INFO: (1/3) Machine panel devID error'. The main area displays three columns of signal status: F (F000-F012), G (G000-G012), and A (A00-A12). Each column has a 'DDRES' and 'VALUE' header. The 'F' column shows values like 01000000, 10000000, etc. The 'G' column shows values like 00000000, 00000000, etc. The 'A' column shows values like 00000000, 00000000, etc. At the bottom, there is a comment line: 'F000 OP SA STL SPL *** *** ***'. The bottom status bar shows 'X.Y.R', 'F.G.A', 'C.T', and a timestamp '09:44:22'.

Fig. 1-18

Note: The comment displayed on this diagnosis page is the current running PLC information. Depending on the PLC, the diagnostic display information may be different, and the currently displayed comment is defined by the ladder diagram editor.

View the status of each signal

X.Y.R

Press the **X.Y.R** soft key to display the status of the X, Y, and R signals in the window; press

F.G.A

the **F.G.A** soft key to display the status of the F, G, and A signals in the window.



Press these buttons to switch between X, Y, R or F,G,A signal bar.



Press these buttons to select and view among the signals of X, Y, F, G and R, A, and T.



Press the **CHANGE** soft key to switch to the position view status, and you can view the status of each signal bit.

1.3.3 PLC data viewing and setting

PLC DATA

Under the ladder diagram page set, press the **PLC DATA** soft key to enter the PLC data status display page, including the setting and saving of K, D, DT, and DC parameters. The displayed page is shown in Fig. 1-19.

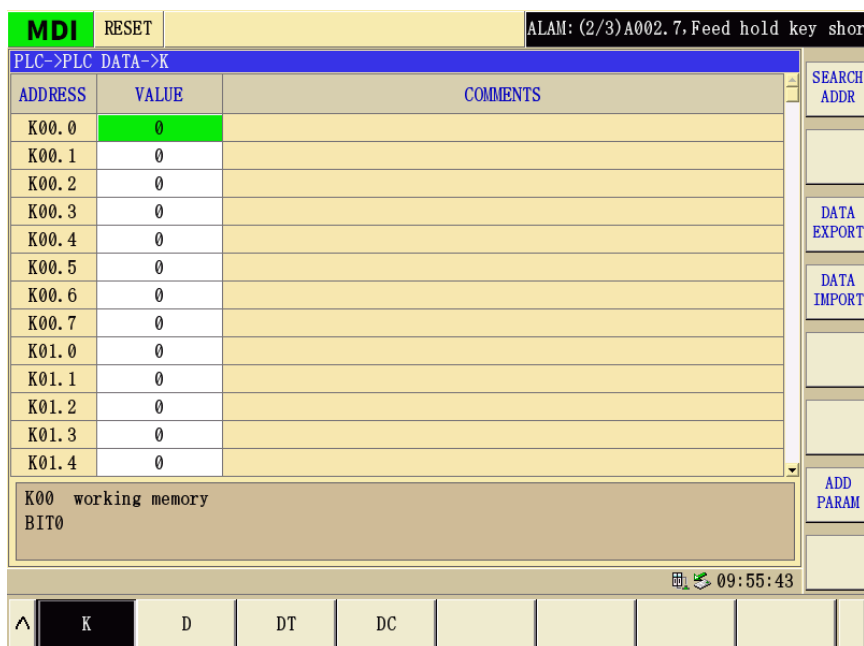


Fig. 1-19

Save PLC data



Use **K**, **D**, **DT**, **DC** to select the type of PLC parameter to be operated.

Hereinafter, "K, D, DT, DC" are collectively referred to as registers. Press the **DATA EXPORT** soft key to save the register data in the current PLC data page to the running PLC; press the **DATA IMPORT** soft key to save the register data of the running PLC to the system; press the **ADD PARAM** soft key to favorite the register data where the cursor is located, which is the same as the favoriting operation for system parameters.

Note 1: When the PLC parameter is modified, the modified value is only saved on the system and not written into the PLC. Therefore, the PLC parameter is not exported when the PLC is exported.

Note 2: The data favorited by pressing the **ADD PARAM** soft key can be viewed in the **SYS** (系統) screen ->



K parameter setting

Under the **PLC DATA** page, press the **K** soft key to enter the K parameter setting display page. As shown in above Fig.:

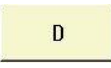
● Parameter Setting Method:

① Press the keys to select the parameter status

bit that needs to be modified; or press  the soft key, input the K variable to be selected, and then press  the soft key to position the cursor on the parameter. The meaning of the status bit is displayed at the bottom of the screen.

- ② In the K variable status bit, press repeatedly to  switch between 0 and 1 to modify the status of the selected K parameter status bit.

D parameter setting

Under the  page, press the  soft key to enter the D parameter setting display page, as shown in Fig. 1-20.

MDI		RESET	2/3)A002.7.Feed hold key short circ	
PLC->PLC DATA->D				
ADDRESS	VALUE	COMMENTS	Min.	Max.
D000	4	total tool position of tool post	1	16
D001	1	X manual axis movable key corresponding	0	6
D002	0	Y manual axis movable key corresponding	0	6
D003	2	Z manual axis movable key corresponding	0	6
D004	0	4th manual axis movable key corresponding	0	6
D005	0	C manual axis movable key corresponding	0	6
D006	0		0	6
D007	1	tool-post selection:0 、Tool row 1, stan	0	6
D008	0	Operation panel selection (0, SOC system	0	2
D009	0			
D010	0	Servo tailstock control shaf	0	6
D011				
D012				
D007 tool-post selection:0 、Tool row 1, standard tool-post 2, GSK servo tool 3, Lio Shing tool-post 4, Delta tool-post				
10:24:14				
^	K	D	DT	DC

Fig. 1-20

● Parameter setting method:

- ① Press , ,  and  keys to select the D parameter to be modified; or press  the soft key to input the D parameter to be selected , and then press the soft key to position the cursor on the parameter. The meaning of this parameter is displayed at the bottom of the screen.
- ② Press the  key to make the selected D parameter in a modifiable state.
- ③ Enter the modified value, and then press  the button to complete the modification.

DT parameter setting

Under the **PLC DATA** page, press **DT** the soft key to enter the DT parameter setting display page.

- **Parameter Setting Method**

Same as D parameter setting.

DC parameter setting

Under the **PLC DATA** page, press **DC** the soft key to enter the DC parameter setting display page.

- **Parameter Setting Method**

Same as D parameter setting.

1.3.4 PLC online editing

In the ladder diagram page set, press **PROGRAM** → **LOCAL** in turn to enter the ladder diagram program directory page, press **↑**, **↓** to select the program to be edited, and press **EDIT** the soft key to enter the edit page. At this time, you can edit the ladder diagram, symbol table, display information table, and initialization data table.

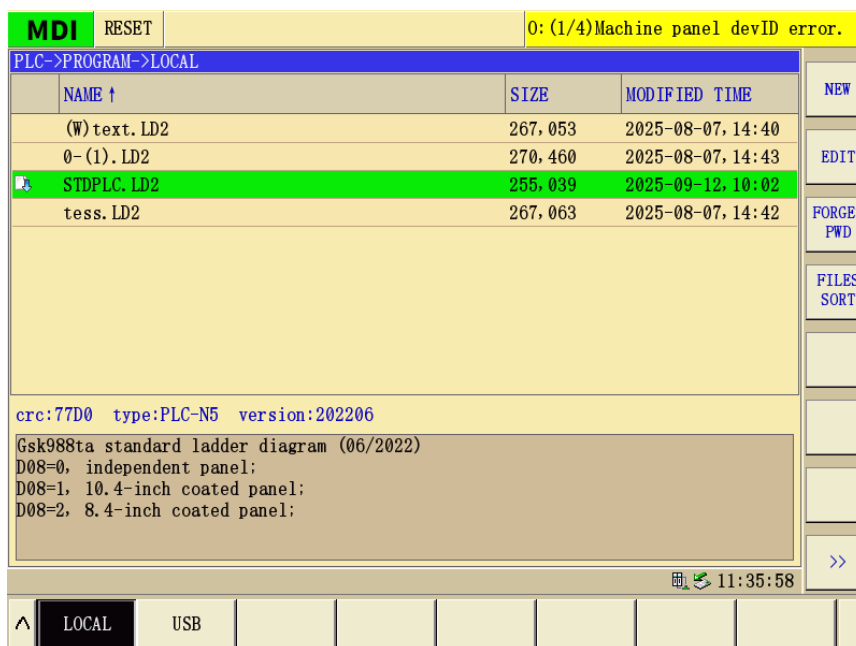


Fig. 1-21

Ladder diagram viewing and editing

In the local directory page, press the **EDIT** soft key to enter the ladder diagram display and edit window display page, as shown in Fig. 1-22.

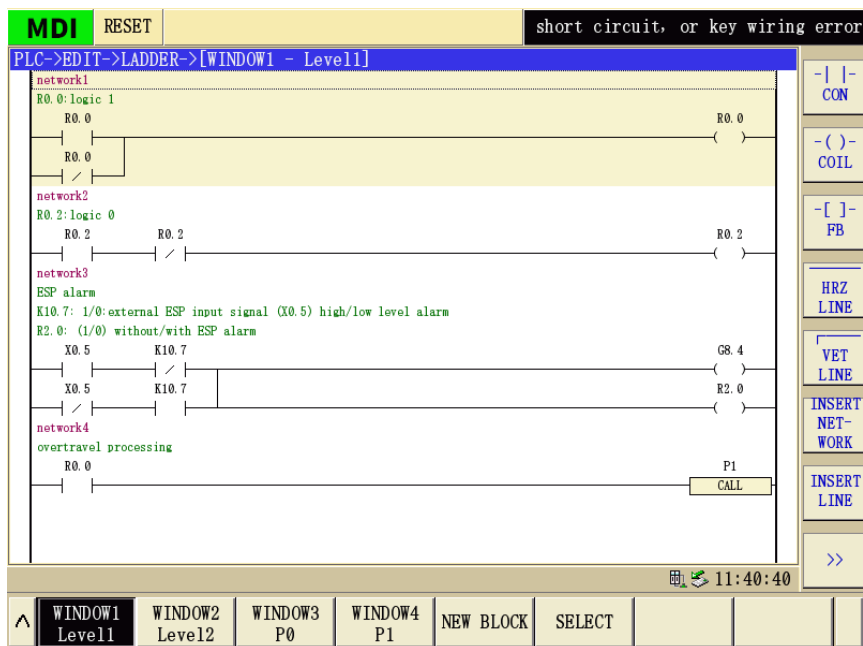


Fig. 1-22

The position of the cursor in the Fig. is indicated by a dashed frame. The background color of the network area where the cursor is currently located is slightly darker than the background color of the window.

Press **WINDOW1 Level1**, **WINDOW2 Level2**, **WINDOW3 P0** and **WINDOW4 P1** corresponding soft keys respectively to display the program blocks corresponding to window 1, window 2, window 3, and window 4 on the screen. The block name of the current window is displayed at the top of the page.

Window block selection:

- ① Press **WINDOW1 Level1**, **WINDOW2 Level2**, **WINDOW3 P0** and **WINDOW4 P1** corresponding soft keys to select

the window that needs to modify the program block. Press the soft key **SELECT** to select the corresponding program block for window 1, window 2, window 3, and window 4.

- ② Use , ,  and  keys to select the corresponding program block in the window, press the **OK** soft key to confirm the selection and return. At this time, the address of the program block corresponding to the window is displayed on the window soft key. For

example **WINDOW1 Level1**, it means that window 1 corresponds to the Level1 program block, that is, when the soft key is pressed **WINDOW1 Level1** at this time, the content of the Level1 program block is displayed in the window.

New program block

- Press the soft keys corresponding to **WINDOW1 Level1**, **WINDOW2 Level2**, **WINDOW3 P0** and **WINDOW4 P1** to select the window where you need to create a new program block.
- Press the **NEW BLOCK** soft key and enter the name of the new block. Press **OK** the soft key to create an empty block for the window.

Edit program

● New editing operation steps:

- First select the window block program to be edited, press **EDIT** the soft key to enter the edit program page (as shown in Fig. 1-23), and press **>>** the soft key to display the extended soft keys (as shown in Fig. 1-24).

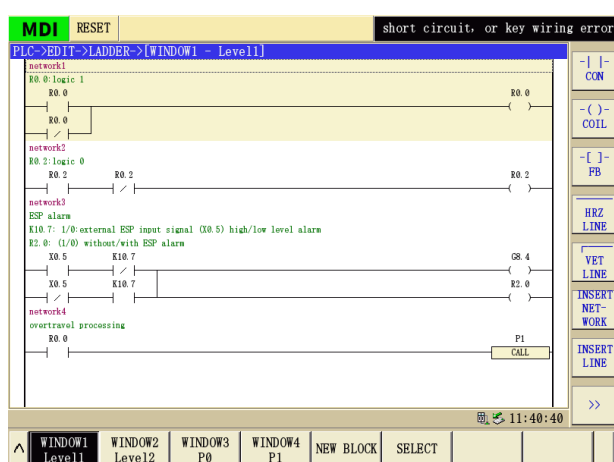


Fig. 1-23

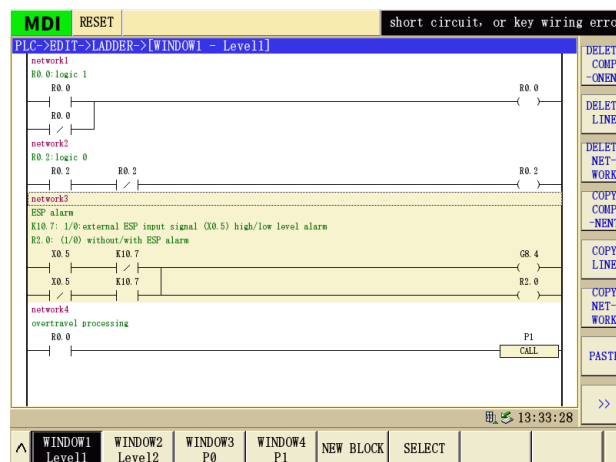


Fig. 1-24

- Press , ,  and  keys to move the cursor to the line that needs to be modified, and press  and  keys to move the cursor to the grid where the element needs to be operated.
- Press the **INSERT NET-WORK** soft key to insert a network before the network where the cursor is located.
- Press the **INSERT LINE** soft key to insert a new line after the line where the cursor is located.
- Edit the PLC program in the new line.

For example:

- Press  the soft key to display the screen as shown in the Fig. below. At this time, the

cursor stops in the "Type" selection box, and press  ,  or  ,  to switch between normally open and normally closed contacts.

- ② Press the  key to switch the cursor to the "address/symbol" edit box, enter the address/symbol, and press the  soft key or the  key to confirm the input.

- ③ Press  the soft key, and the operation is  the same as the soft key.

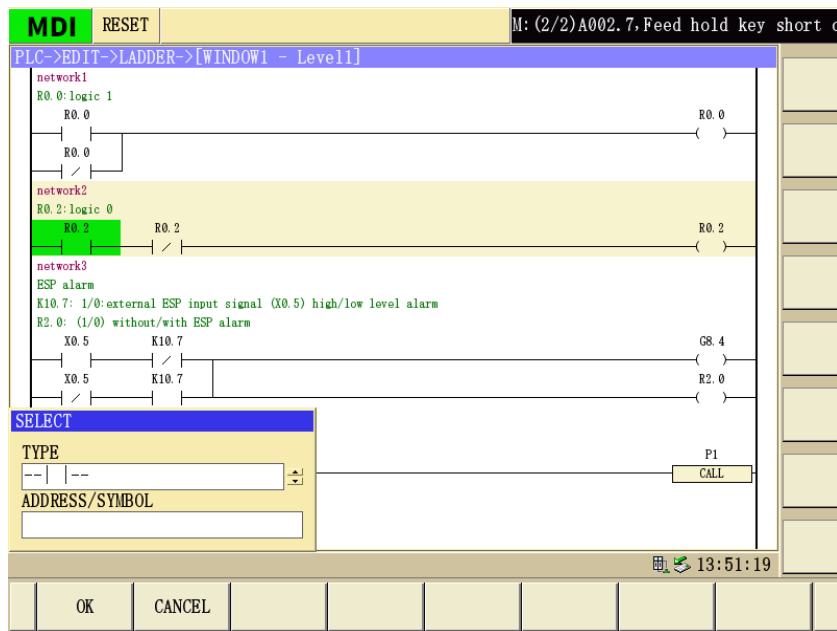


Fig. 1-25

- ④ Press the  soft key to display the screen as shown in Fig. 1-26.

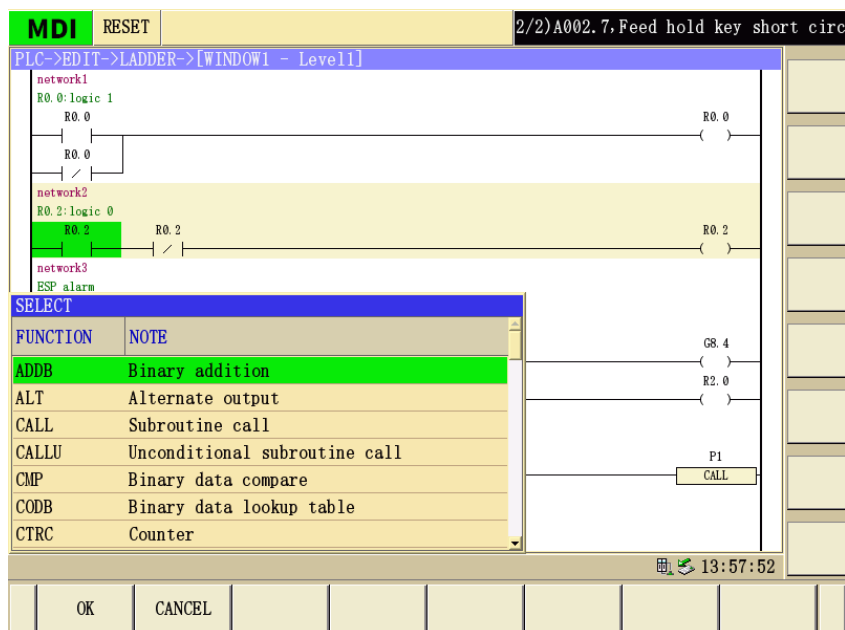


Fig. 1-26

- ⑤ Press  and  keys to select the function instruction to be inserted, as shown in Fig. 1-26 ADDB (binary addition), and press the  soft key or  key to display the page as shown in Fig. 1-27.

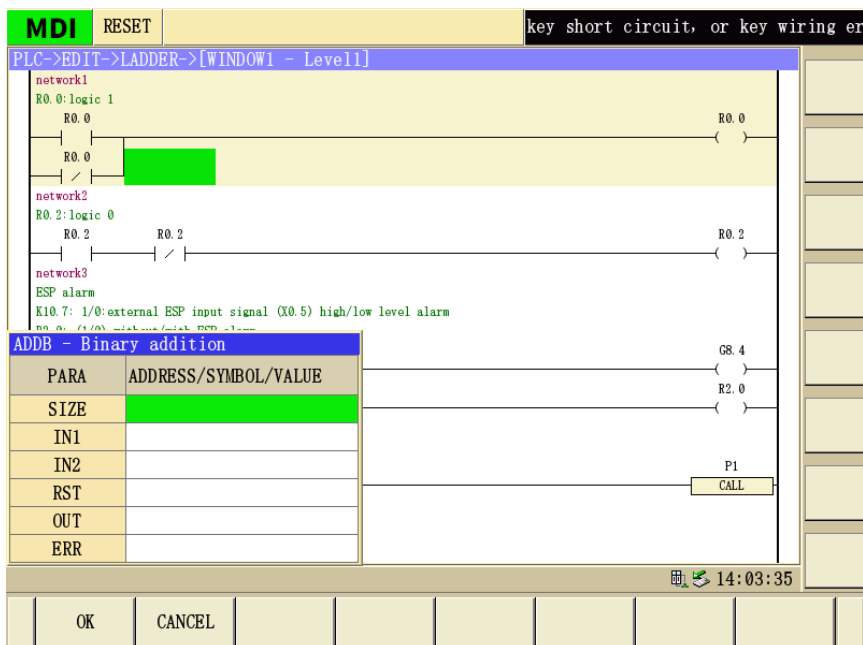


Fig. 1-27

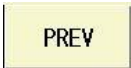
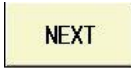
- ⑥ Press  and  keys to switch up  and down in each edit box to select the edit box, enter the address or data, and press the key to confirm the modification. After all the

edit boxes are entered, press the  soft key to finish editing.

- ⑦ The input and editing operations of other function commands are the same as described above.

● **Other operations of the PLC program:**

In addition to the above-mentioned program editing functions, you can also delete, copy, paste and other editing operations on the selected component, line or network. You can also undo the last operation or redo the last undone operation.

- ① Press the  soft key to enter the search page, enter the address to be searched in the dialog box and use ,  and the cursor jumps to the first address position that is searched forward or backward based on the current page position.

- ② Press , ,  soft keys to delete the line, network or component at the cursor position.

- ③ Press the , ,  soft keys to copy the line, network or component at the cursor position. After copying is completed, press  and  keys to move the cursor to the network, line and component to be operated, and press  the soft key to paste the copied line, network or component.

- ④ Press the  soft key to undo the operation of the previous step, up to 20 steps can be memorized.

- ⑤ Press the  soft key to restore the previous undo operation.

Symbol table viewing and setting

In the ladder diagram edit page, press the  soft key to enter the symbol table display page, as shown in Fig. 1-28.

MDI

RESET

INFO: (1/2)Machine panel devID error

PLC->EDIT->SYMBOL TABLE->WINDOW1(P)

	SYMBOL	ADDRESS	COMMENTS
1		P0000	Initial_Power_On
2		P0001	Over_Travel_Signal
3		P0002	Machine_Panel_Input_Signal_Map
4		P0003	Machine_Panel_Output_Signal_Map
5		P0004	MPG_trial_cut
6		P0005	Work_Mode_Main
7		P0006	Work_Mode_Key
8		P0007	Jog_Main
9		P0008	Jog_Move_Key
10		P0009	MPG_Main
11		P0010	MPG_Axis_Choose_Key
12		P0011	MPG_Override
13		P0012	Jog_Return
14		P0013	External_MPG_Iput
15		P0014	Feedrate_Override_Ajust
16		P0015	Rapid_Traverse_Main

SELECT

SEARCH
ADDR

INSERT
LINE

DELETE
LINE

14:19:03

WINDOW1
P

WINDOW2
X

WINDOW3
Y

WINDOW4
K

Fig. 1-28

- ① Press **WINDOW1 P**, **WINDOW2 X**, **WINDOW3 Y** and **WINDOW4 K** soft keys respectively to select the symbol table information to be displayed on the screen. At this time, the name displayed in the current window and the corresponding symbol table name are displayed at the top of the window.

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

The above Fig. shows **WINDOW1 P** the program block symbol table, and the program block symbol table displays the address table corresponding to the program symbol.

- ② Press to **WINDOW2 X** display the X table, as shown in Fig. 1-29.

MDI

RESET

)A002.7.Feed hold key short circuit

PLC->EDIT->SYMBOL TABLE->WINDOW2(X)

	SYMBOL	ADDRESS	COMMENTS
1		X0	G37 G36 ESP G31 *** TOOL *** **
2		X0.0	spare
3		X0.1	spare
4		X0.2	Manual tool setting signal
5		X0.3	spare
6		X0.4	G31 jump signal
7		X0.5	ESP
8		X0.6	G36 jump signal
9		X0.7	G37 jump signal
10		X100	*** PRES *** DITW *** DIQP SHUT OPEN
11		X100.0	Opening detection signal of protective door
12		X100.1	Protection door closing detection signal
13		X100.2	chuck input signal
14		X100.3	spare
15		X100.4	tailstock control signal
16		X100.5	spare

SELECT
SEARCH
ADDR
INSERT
LINE
DELETE
LINE

14:26:10

WINDOW1
P

WINDOW2
X

WINDOW3
Y

WINDOW4
K

Fig. 1-29

The X table shows the annotation meaning of X, Y, DC, DT, T, R and other parameter addresses.

Use , ,  and  keys to select and view each parameter address.

Press the  soft key, input the address of the parameter to be searched, and position the cursor at the address.

Press the  soft key to insert a blank line below the line where the cursor is located.

Press the  soft key to delete the line where the cursor is currently located.

③ ,  the interface and operation are  the same.

Modify and Edit Symbol Table (Program Block Symbol Table cannot be Modified Here)

Select the window symbol table to be modified, use , , , , , 

keys to select the symbol, address or comment column to be modified , press the key to make the selected column into the input state, enter the symbol, address or comment, and then press  the key to complete the modification .

View and modify display information table

In the edit page, press the  soft key to enter the display information table display page, as shown in Fig. 1-30.

MDI		RESET	ALAM: (2/2)A002.7.Feed hold key shor	
PLC->EDIT->MESSAGE TABLE->WINDOW1				
ADDRESS	MSG NO.	DISPLAY CONTENT		
A0000.0	1000	Excessive tool change time		
A0000.1	1001	Current tool-position is inconsistent with the object one wher		
A0000.2	1002	Tool change does not complete		
A0000.3	1003	Tailstock function invalid.M10/M11 commands can not be perform		
A0000.4	1004	Tailstock can not be withdrawn when spindle rotates.		
A0000.5	1005	Automatic door opening and closing timeout		
A0000.6	1006	The safety door does not close. the machining program/spindle		
A0000.7	1007	Low pressure of hydraulic station		
A0001.0	1008	Do not loose the chuck when spindle rotates.		
A0001.1	1009	Spindle can not be started up if the chuck clamping is not ger		
A0001.2	1010	Chuck clamping signal is not detected when the spindle is rota		
A0001.3	1011	Spindle can not be started up if the chuck is released.		
A0001.4	1012	Chuck function can not being performed M12/M13 command. due to		
A0001.5	1013	Tool post lock signal is not detected when tool change is ende		
A0001.6	1014	instruction tool number is 0		
A0001.7	1015	do not allow chuck action when the spindle is running		
14:50:11				
^	PLC	SYMBOL	MSG TABLE	DATA TABLE
			DIAG TABLE	MCODE TABLE
			LEVEL TABLE	IO MAP
				>

Fig. 1-30

The display information table shows the PLC alarm information A address, the corresponding information number and the corresponding displayed information content. Press , , , , ,  keys to select and view each address, information number and its corresponding display information.

① Modification of information number and display content

Press , , , , ,  keys to select the information number or display content to be modified, press  the key to make the selected information number or display content in a modifiable state, enter the information number or display content to be modified, and then press  the key to complete the modification.

② Address, information number lookup

Press the  or  soft key, input the address or information number to be searched, press the OK key to search and locate the cursor to the address or information number found.

Data sheet viewing and setting

In the edit page, press the  soft key to enter the initialization data table display page, as shown in Fig. 1-31.

MDI		RESET		short circuit, or key wiring error					
PLC->EDIT->INITDATA TABLE->WINDOW1(K)									SELECT
ADDRESS	7	6	5	4	3	2	1	0	SEARCH ADDR
K00	0	0	0	0	0	0	0	0	
K01	0	0	0	0	0	0	0	0	
K02	0	0	0	0	0	0	1	0	
K03	0	0	0	0	1	1	0	0	
K04	0	0	0	0	0	0	0	0	
K05	0	0	0	0	0	0	0	0	
K06	0	0	0	0	0	0	0	0	
K07	0	0	0	0	0	0	0	0	
K08	0	0	0	0	0	0	0	0	
K09	0	0	0	0	0	0	0	0	
K10	1	0	0	0	0	0	0	0	
K11	0	0	0	1	0	1	0	0	
K12	0	0	0	0	0	0	0	0	
K03 SLC2 SLC1 ROV2 ROV1 *** *** *** ***									
BIT2									
15:01:17									
WINDOW1 K	WINDOW2 D	WINDOW3 D FLOAT	WINDOW4 DT						

Fig. 1-31

① K parameter setting

Press  the selection window 1, the corresponding display is the detailed information of the

K parameter, as shown in the Fig. above:



Press the , , , , and keys to select a digit corresponding to the K parameter that needs to be set or modified. The bottom of the display page window shows the specific meaning of the corresponding digit of the K parameter where the cursor is currently located.



Press the key repeatedly to switch the selected bit between 0 and 1.

② Initialize data



Press the soft key to enter the page for selecting the initialization data table, as shown in Figure 1-32.

MDI

RESET

short circuit, or key wiring error

PLC->EDIT->INITDATA TABLE->WINDOW2 (D)

NO.	ADDRESS	VALUE	Min.	Max.
1	D0	4	1	16
2	D1	1	0	6
3	D2	0	0	6
4	D3	2	0	6
5	D4	0	0	6
6	D5	0	0	6
7	D6	0	0	6

SELECT INITDATA TABLE

TALBE NAME

D

D_FLOAT

DT

DC

post

15:03:20

OK

CANCEL

Fig. 1-32

③ Revision and editing of data table:



Press the page up key and the page down key to select the desired page, the arrow



keys, , , and select the address value to be modified or the value, maximum and minimum value of an address, the selected value turns into a green background, press



the key to make the value In the modifiable state, use the numeric keys and the backspace key



to modify the value data, and then press the key to confirm the modification.

1.3.5 PLC program transmission

PLC program transmission can only be carried out if the authority is above the machine manufacturer level (level 2).

There are two ways to transfer PLC programs:

1. Use GSKEdit for transmission.
2. Use U disk to transfer a single PLC program on the ladder diagram page, or perform batch PLC program transfer on the file management page. For detailed operations, please refer to "1.2 U Disk Operation" in this chapter.

1.4 System debugging operation

In order to facilitate the user to debug the machine tool, the CNC system classifies the commonly used parameters, which is called the system debugging function.

Note: There may be inconsistencies in the parameter classification and content of the system debugging function between different system software versions. Please refer to the currently used version.



Press the function key to enter the setting page set; in the setting page set, press the SYSTEM ADJUST soft key to enter the system debugging page, as shown in Fig. 1-33.

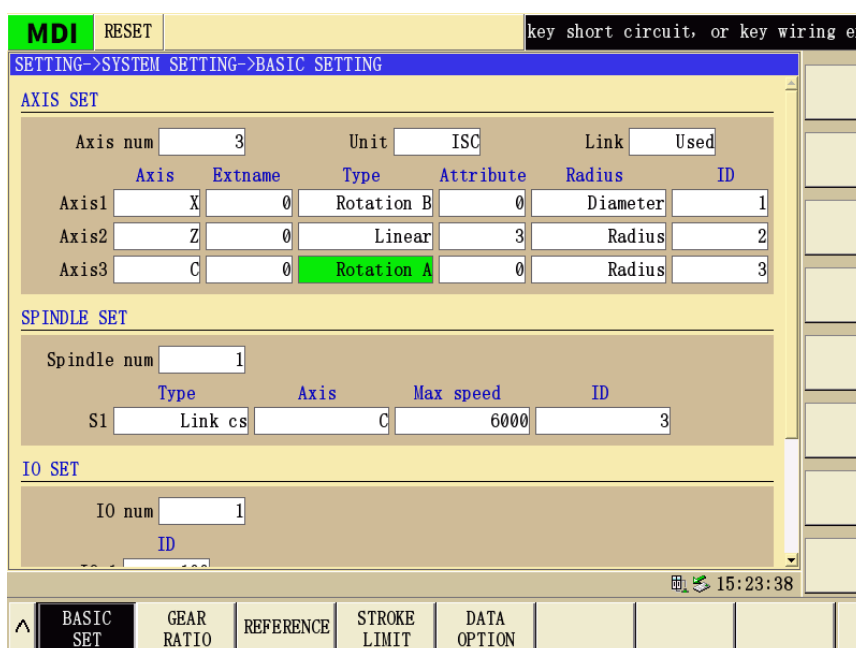


Fig. 1-33

After the system is connected to the servo unit, I/O unit or gateway, the relevant parameters need to be set to carry out normal bus communication. The system debugging page categorizes the parameters according to functions, and the user completes the parameter settings in order according to the prompts on the page.

The debugging items are as follows.

Serial Number	Function Name	Introduction
1	Basic Settings	Set the number of feed axes, the number of spindles, the name of the feed axis, linear/rotary axis settings, feed axis output direction, Cs axis settings, I/O unit analog spindle settings, additional axis settings, etc.
2	Bus Settings	GSKLink slave station settings (each slave station can correspond to the following CNC side settings: feed axis, spindle, I/O unit, gateway)
3	Gear ratio setting	Automatically calculate the servo gear ratio according to the encoder line number and screw lead data
4	Reference point settings	Select incremental/absolute reference point approach, set soft limit
5	Backup/restore Data	Perform one-key backup and one-key restore operations

Note: After debugging, you can use the "one-key backup" function to save the modified system parameters, PLC programs, PLC parameters, and servo parameters; the backed-up data can be restored to this CNC or other CNCs through the "one-key restore" function .

1.4.1 Preparation for commissioning

1. Check the working mode, running status, parameter switch, etc.

	Condition	Descriptions
Operating Status	Reset state	
Operation Mode	MDI	
Permission Settings	Machine tool manufacturer level	Some parameters require level 2 permissions
Parameter Switch	On	
Parameter	No.9000.0 is 1	Whether the system GSK-Link communication function valid

2. Enter the debugging wizard interface, and press on the editing panel in  →  turn to enter the setting page, as shown in Fig. 1-34.

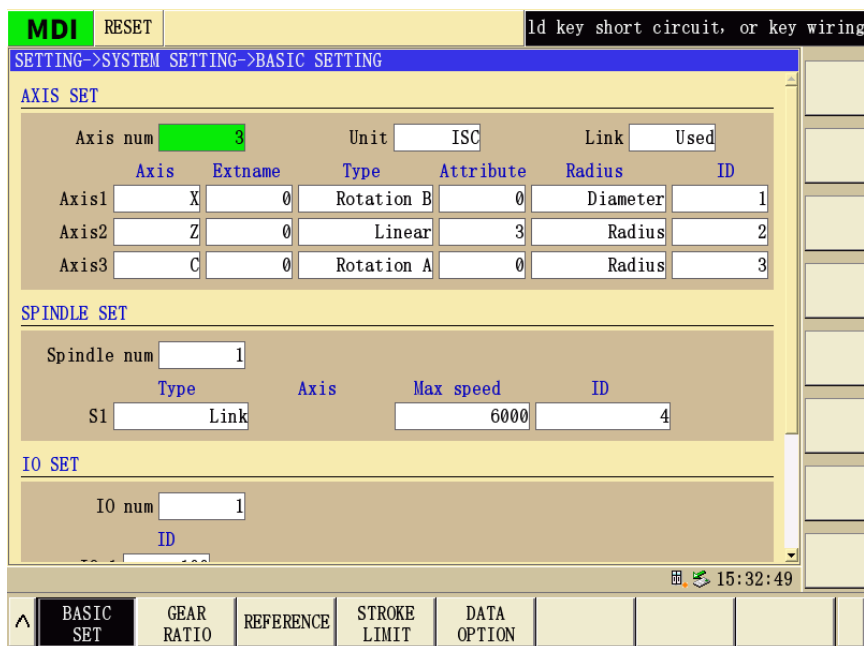


Fig. 1-34

1.4.2 Basic positions

Press → → to enter the setting page.

● Feed Axis Setting

Press → → to enter the basic setting page.

- Set the type of feed axis (linear axis, rotary axis) and axis attributes:

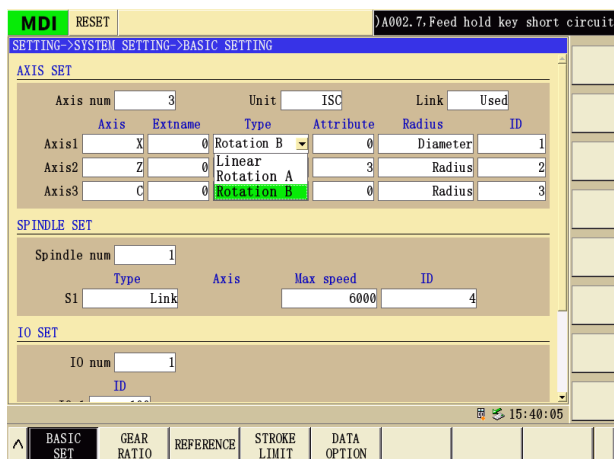


Fig. 1-35

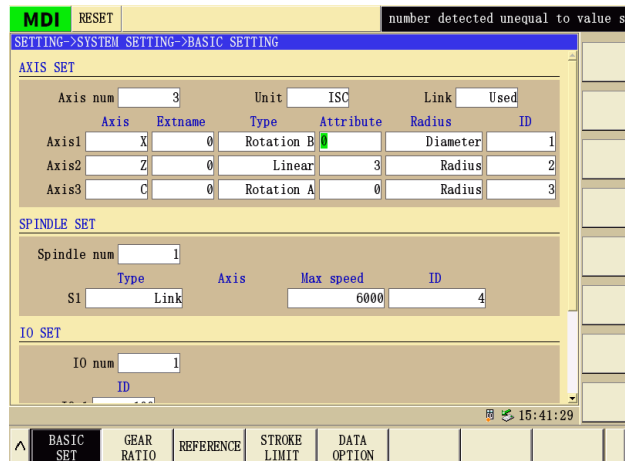


Fig. 1-36

- Set the straight radius and station address of the feed axis:

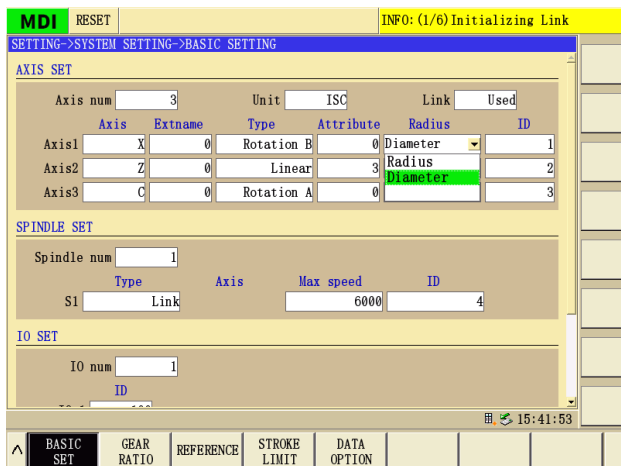


Fig. 1-37

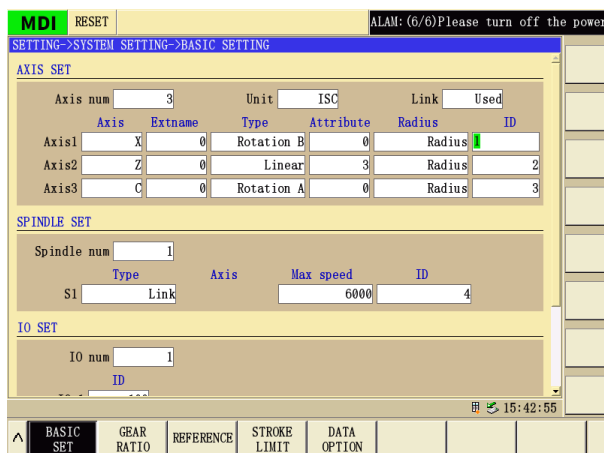


Fig. 1-38

- Set the axis number and minimum unit:

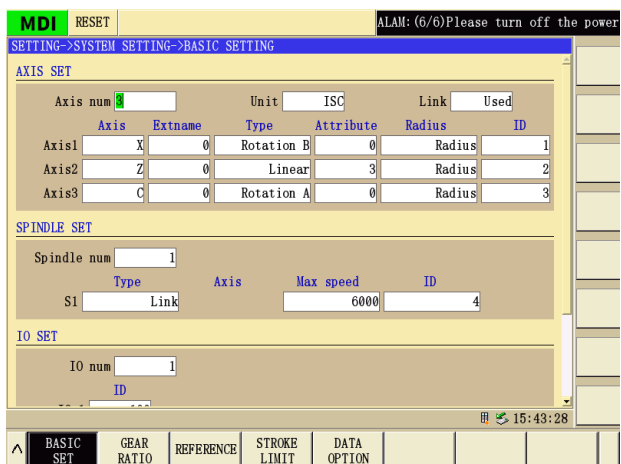


Fig. 1-39

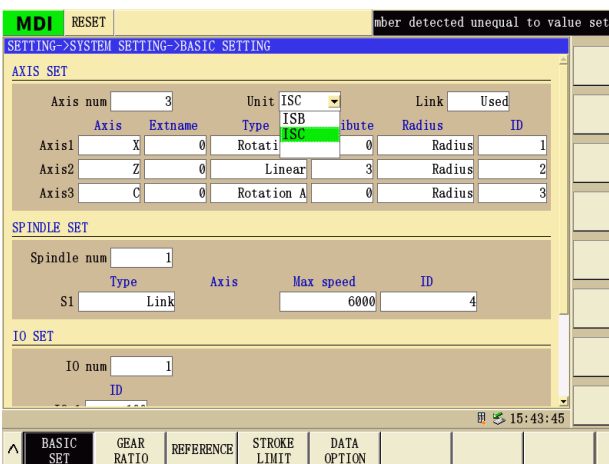


Fig. 1-40

- Set the bus function of the feed axis and feed axis name:

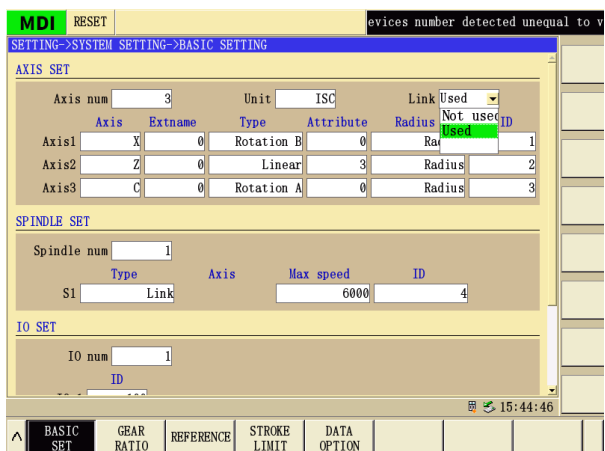


Fig. 1-41

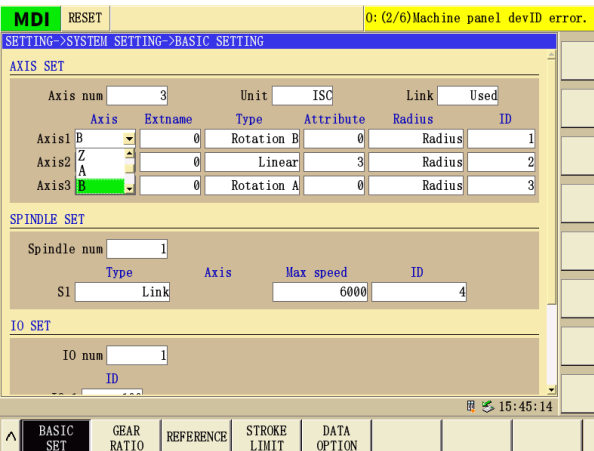


Fig. 1-42

Note: The "axis name" here is set by the parameter 1020, and the "extname" is set by the parameter 1025. Extname means that when the axis name is set in 1020, if two X-axes may be set, a subscript can be set in the 1025 parameter to distinguish the

two axes. For the axis without subscript, use it in the normal way, for example: G1 X100. For the axis name with subscript set (parameter 1025 setting value is not 0), you need to use the axis name plus subscript method when you want to use the axis. If one of the two X axes is set to 1, the parameter No. 1025 can be used like this Use: G1 X1=100.

- **Spindle settings**

- Modify the number of spindles and spindle type:

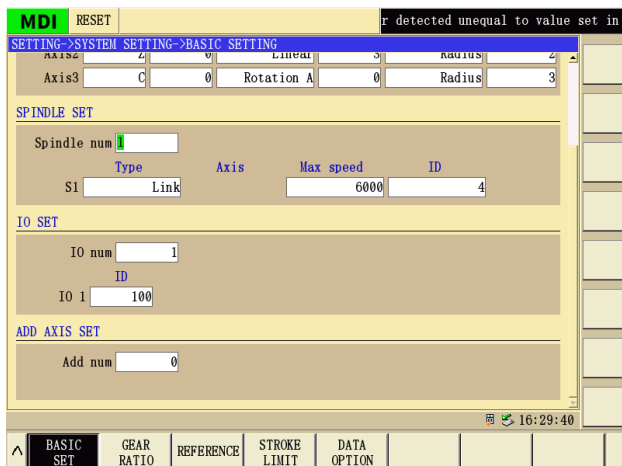


Fig. 1-43

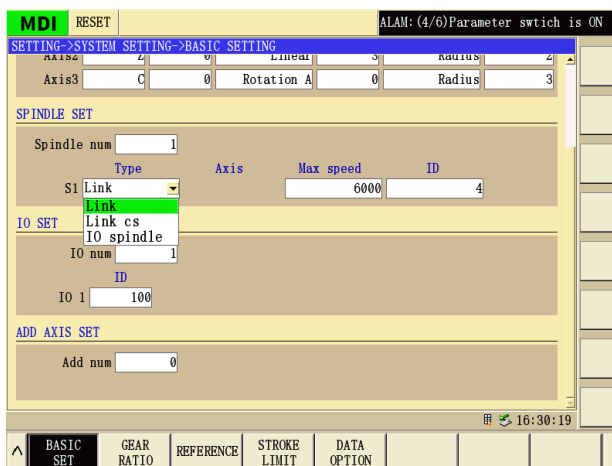


Fig. 1-44

- When setting as the bus Cs axis, you need to select the corresponding feed axis name:

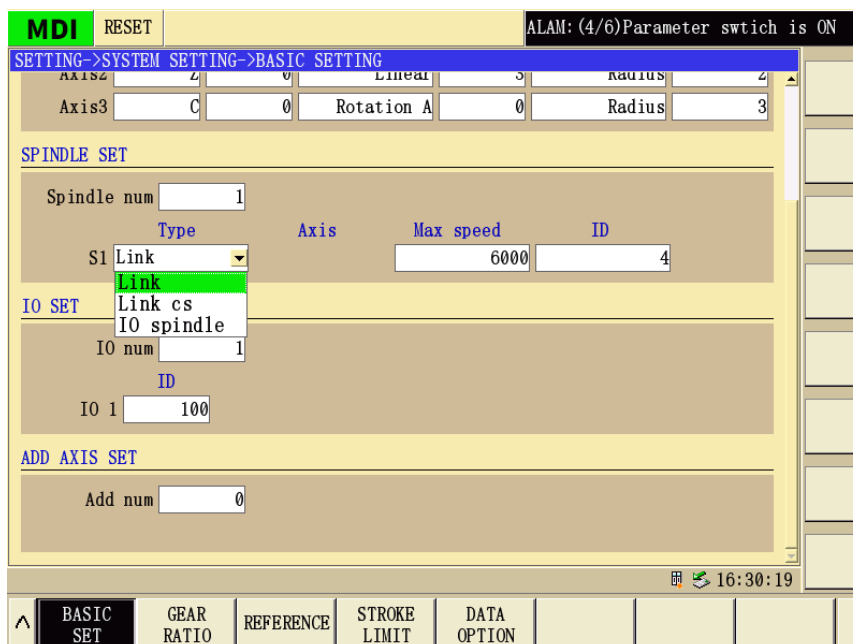


Fig. 1-45

Note: After setting the axis name of the Cs axis (such as the C axis in this example), the type of the C axis is automatically set to A-type rotary axis, and the axis attribute is also set to 0.

- Set the spindle as an I/O analog spindle (output through the analog output port of the I/O unit):

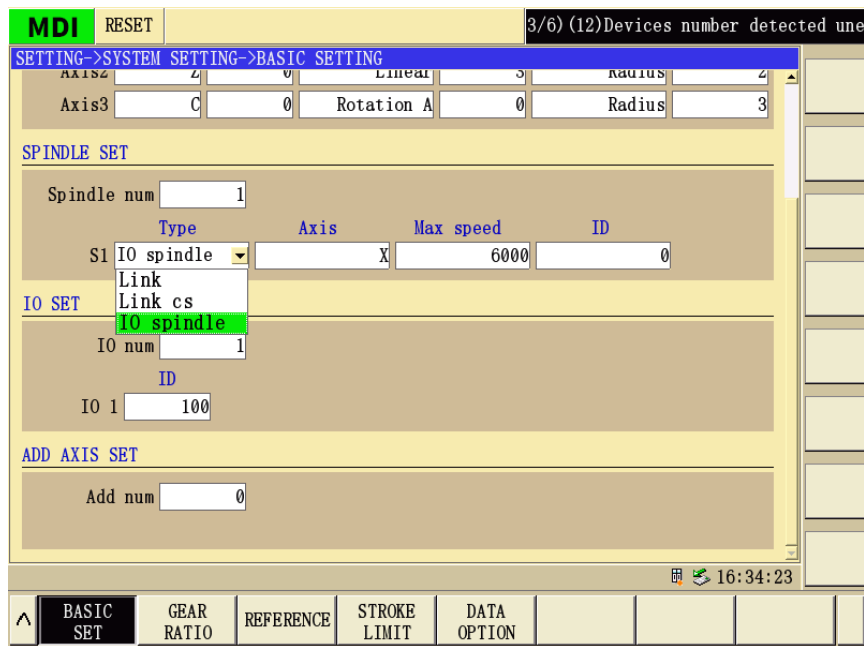


Fig. 1-46

Note: The I/O analog spindle uses the analog output AOT0 of I/O unit 1 by default. If 2 I/O analog spindles are set, the analog output AOT1 of I/O unit 1 is used. If 3 I/Os are set to simulate the spindle, use the analog output AOT2 of I/O unit 1.

■ Set the encoder channel of the spindle

In general, the encoder channel defaults to 0, which means that the spindle type is a bus spindle. If it is an analog spindle, to obtain the spindle speed through an external encoder, you need to select the encoder channel. Parameter 3723 is used to set the encoder channel of the spindle.

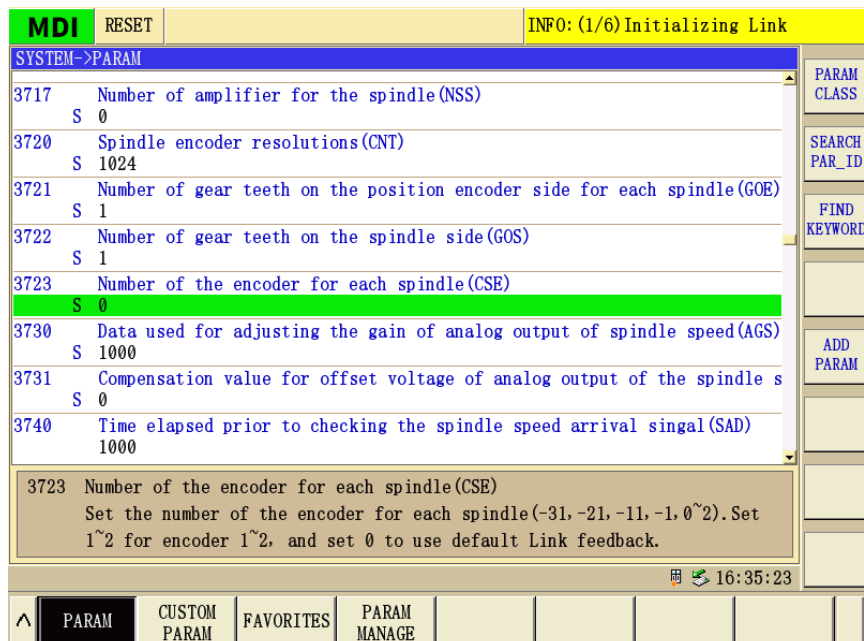


Fig. 1-47

		Parameter Number		Note
Feed Axis Setting	Number of Feed Axis	1010		CNC control axis
		8130		Total number of control axes
	Axis Name	1020		Programming name of each axis
	2 nd Axis Name	1025		The second programming name of each axis
	Type	1006	BIT0	00: Linear axis 01, 11: Rotary axis
			BIT1	
	Axis Attributes	1022		Attributes of each axis in the basic coordinate system
Spindle Setting	Number of Spindles	3710		CNC control spindle number
	Type	3701	BIT7	The spindle uses Cs contour control
	Feed Axis	3717		Amplifier number of each spindle
	Encoder Channel	3723		Channel number corresponding to each spindle encoder

- Set the station address and maximum rotation speed:

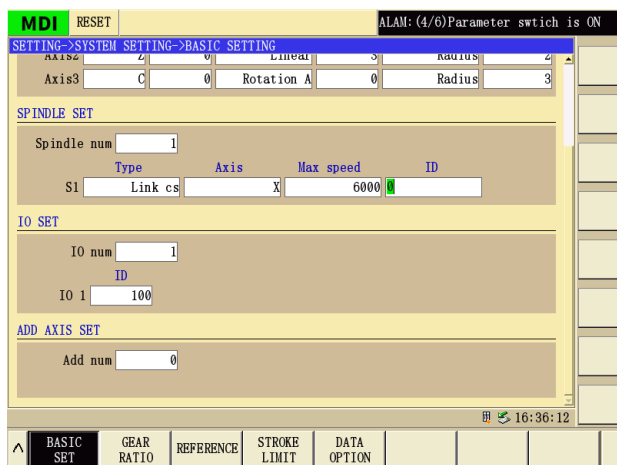


Fig. 1-48

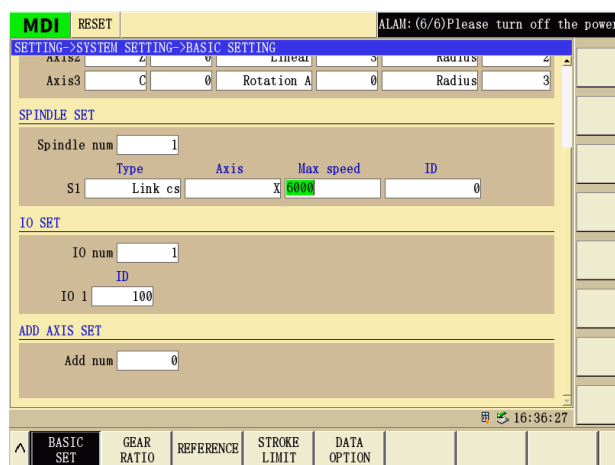


Fig. 1-49

- I/O unit settings
 - Set the number of I/O units and station address:

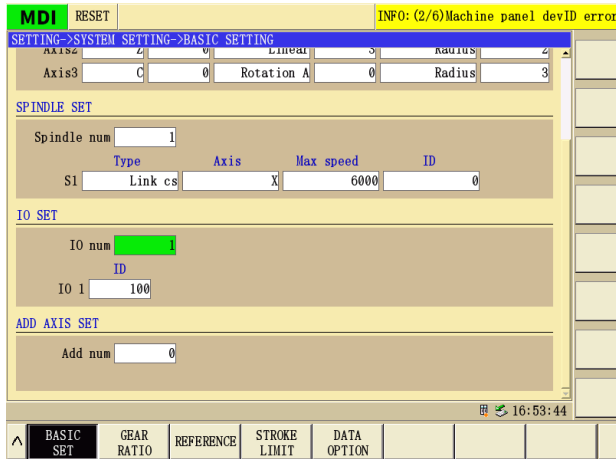


Fig. 1-50

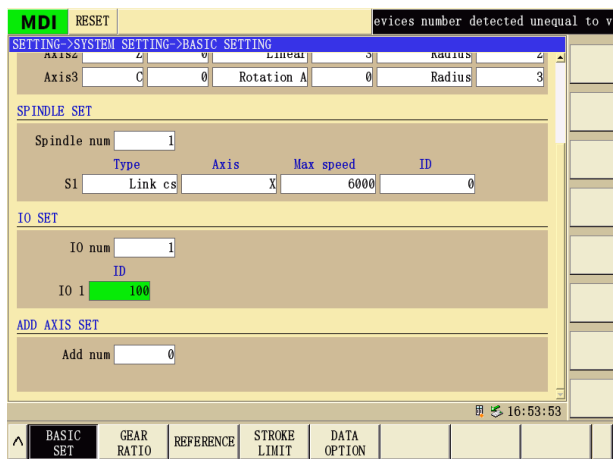


Fig. 1-51

● Additional axis settings

- Set the total number of control axes for additional axes:

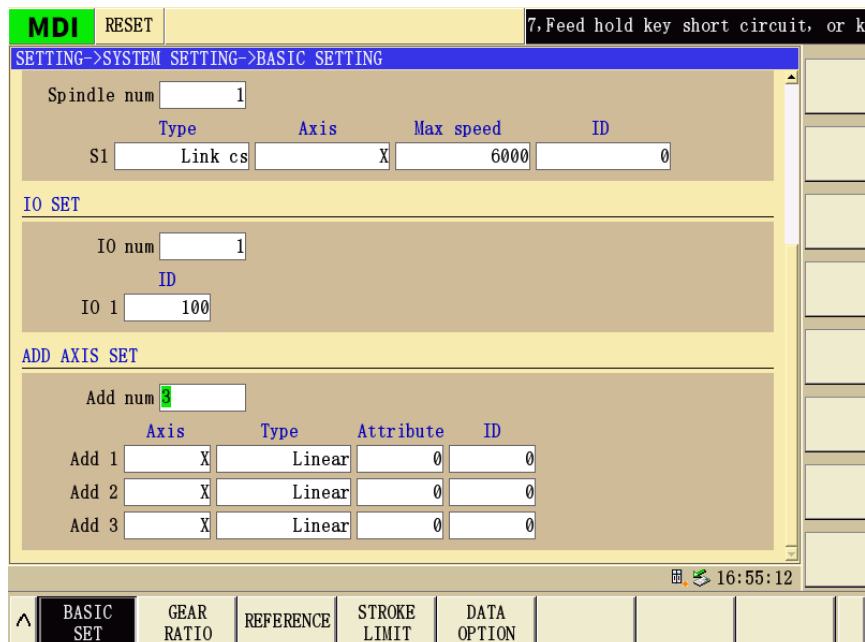


Fig. 1-52

- Set additional axis name and additional axis type:

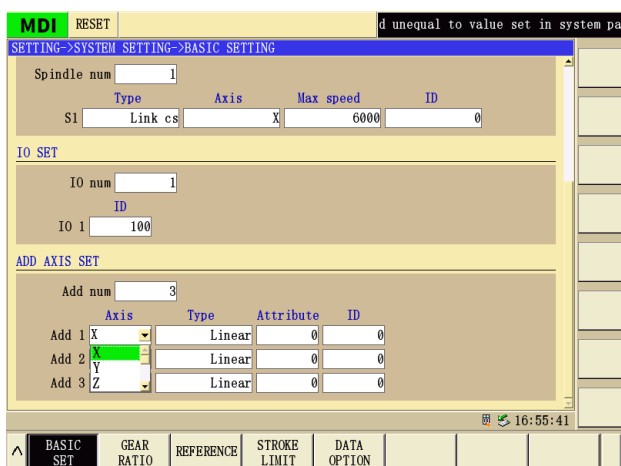


Fig. 1-53

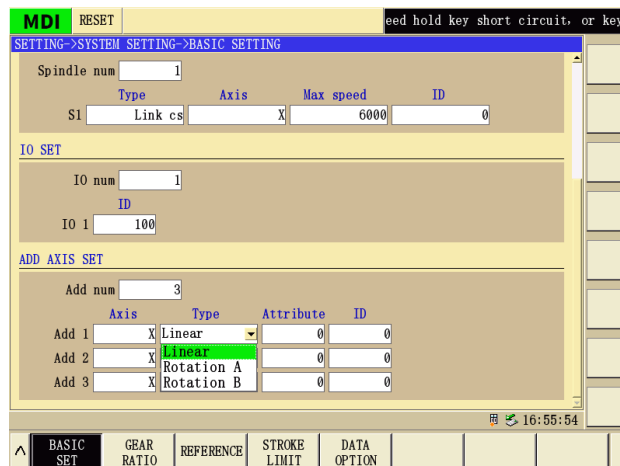


Fig. 1-54

- Set additional axis attributes and station address:

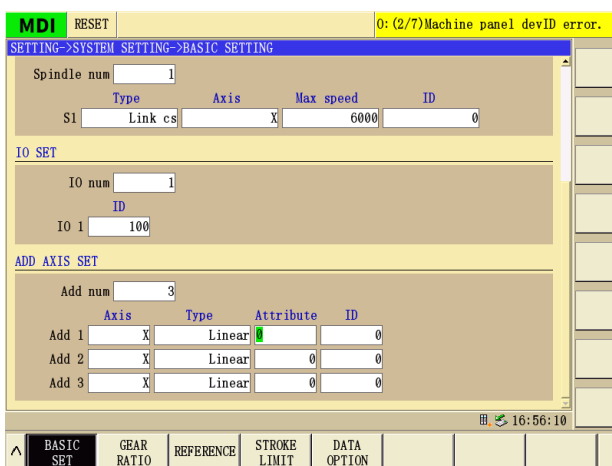


Fig. 1-55

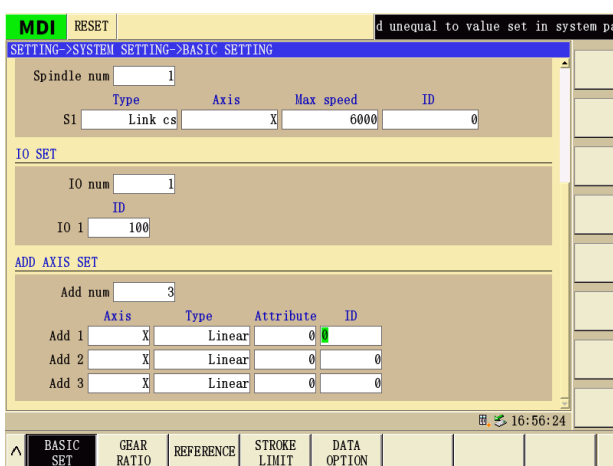
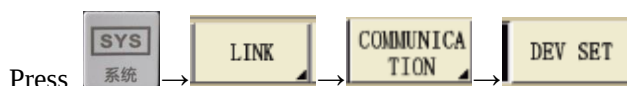


Fig. 1-56

1.4.3 Bus settings

The GSK-Link bus can connect the servo unit, I/O unit and gateway device, and the connection sequence between the devices can be arranged arbitrarily, and the control is not affected by the arrangement. First, set the parameter No.9000.0 to '1' and turn on the GSK-Link bus communication switch.



Press **SYS** (系统) → **LINK** → **COMMUNICATION** → **DEV SET** to enter the setting page. The page displays the configuration of each slave station in turn according to the actual connection order.

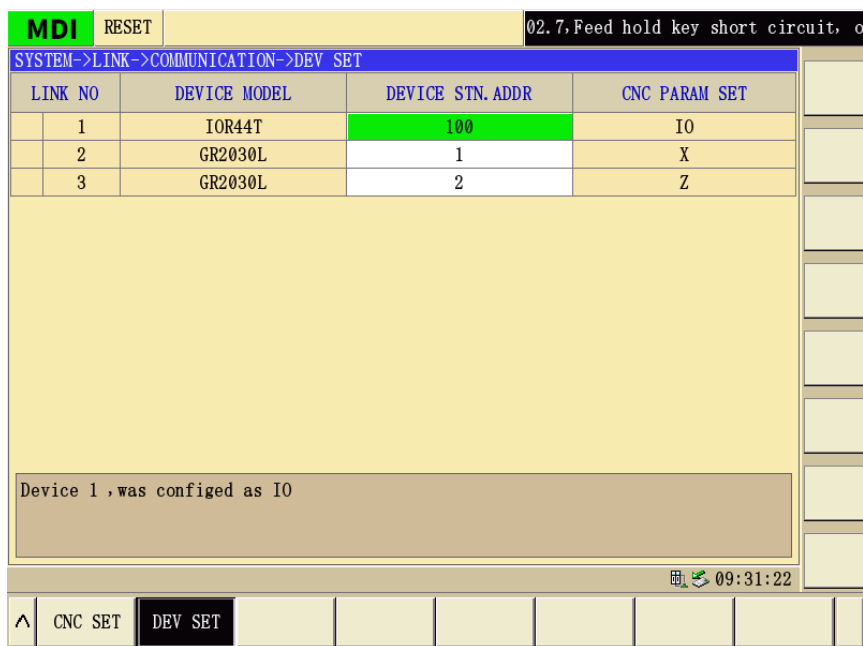


Fig. 1-57

If it is the first power-on, the slave station may not correspond to any system settings, and GSK-Link has detected an error at this time.

After setting the correct number of feed axes and spindles, all slave stations are listed according to the GSK-Link connection configuration. The user can select the system settings of each slave station according to the actual application. The example is as follows.

Slave Station Number	Corresponding System Settings
Slave Station 1	X-axis
Slave Station 2	Z-axis
Slave Station 3	I/O unit 1
Slave Station 4	Gateway 1

Note: The serial number of the slave station is obtained according to the wiring sequence, from the GSK-Link B interface of the CNC to the GSK-Link A interface, the slave station 1, the slave station 2, ..., and so on.

The station address of the connected device can be set on the Device Settings page, as shown in Figure 1-58.

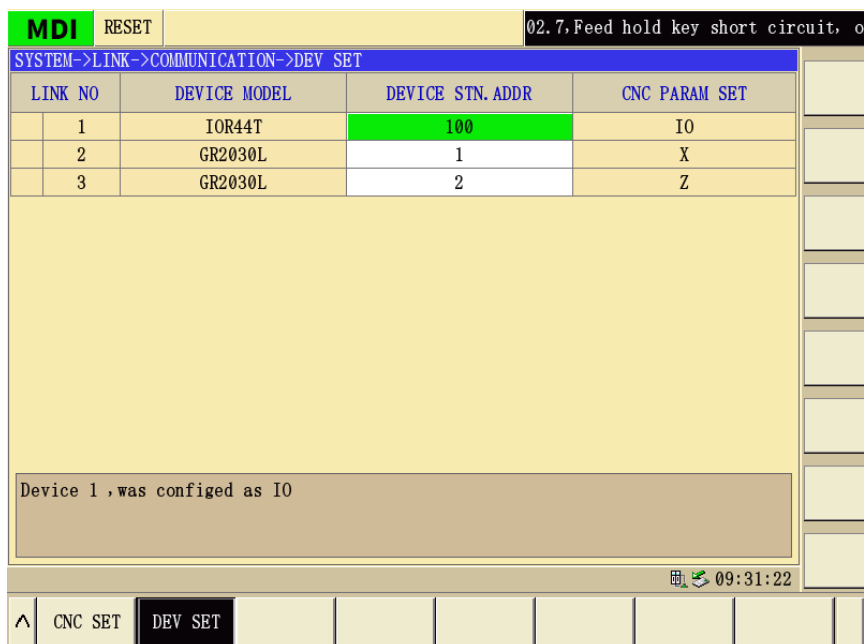


Fig. 1-58

The station address of the feed servo slave station can be set on the System Settings page, as shown in Figure 1-59.

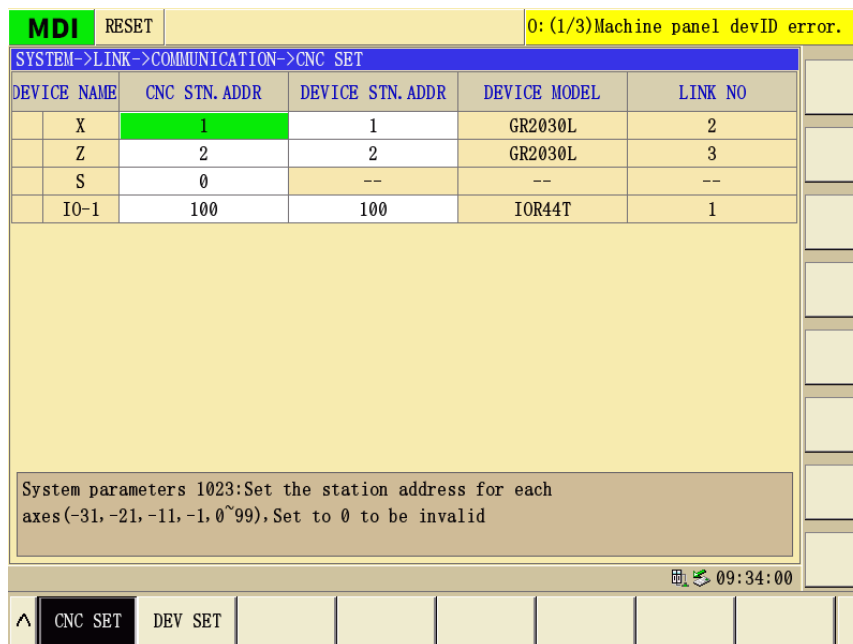


Fig. 1-59

If the corresponding system settings are incorrect, the system will trigger an alarm. Please perform the correct settings according to the alarm information.

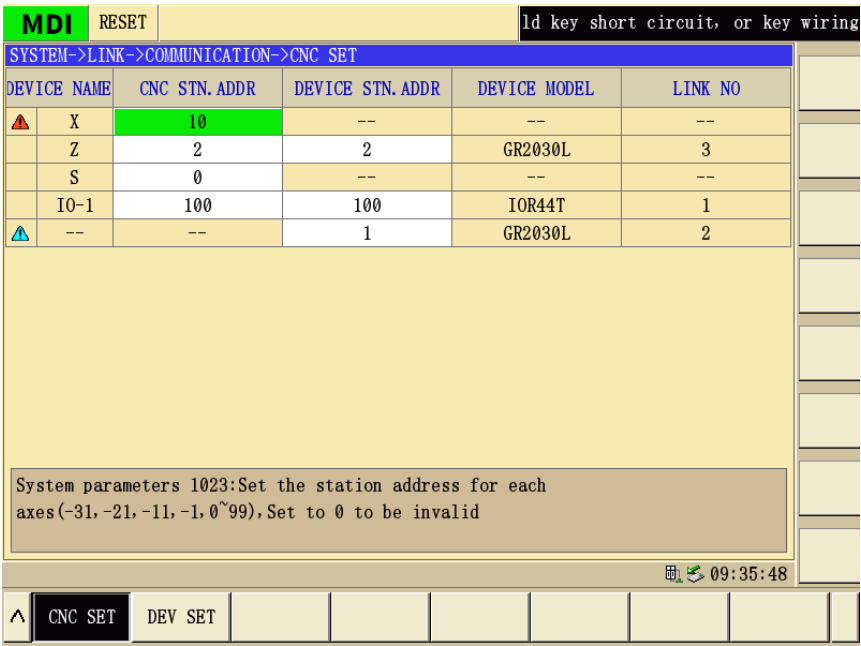


Fig. 1-60

	Parameter Number	Note
Corresponding system parameter settings	1023	Station address of each axis
	3717	Amplifier number of each spindle
	3050	Number of I/O units
	3051	Station address of I/O unit 1
	3052	Station address of I/O unit 2
	3053	Station address of I/O unit 3
	3054	Station address of I/O unit 4
	3060	Number of gateways
	3061	Station address of gateway 1
	3062	Station address of gateway 2

1.4.4 Gear ratio setting

Press  →  →  in turn to enter the setting page, as shown in Fig. 1-61.

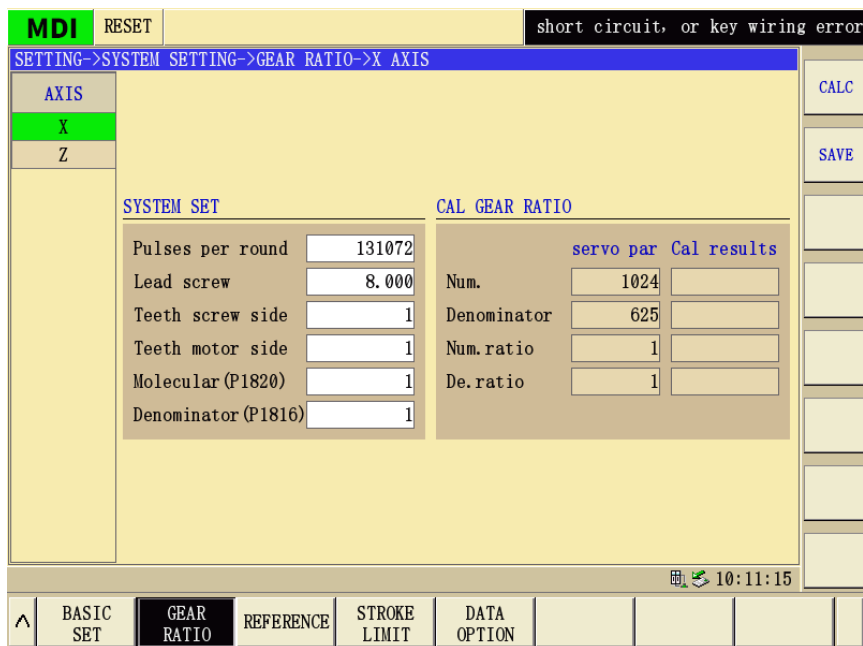
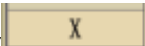
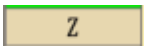
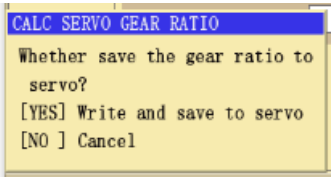


Fig. 1-61

	Corresponding Parameter Number	Note
Encoder 1 revolution pulse		Automatically read from the servo after power-on, and can also be modified manually
Lead screw		Manual input, 360 rotation axis input
Number of gears at the screw end		Transmission ratio numerator
Number of gears at motor end		Transmission ratio denominator
Command multiplication Ratio	1820	
Detection multiplication ratio	1816	

Press the soft key , , ... to switch the display among each feed axis. Press  the soft key for the calculation result. If the displayed axis is enabled with GSK-Link communication function, it will

be displayed . A dialog box pops up , press  the soft key to save the calculation result to the servo parameter PA29/PA30.

1.4.5 Reference point setting

1.4.5.1 Set reference point

Press  →  →  to enter the setting page, as shown in Fig. 1-62.

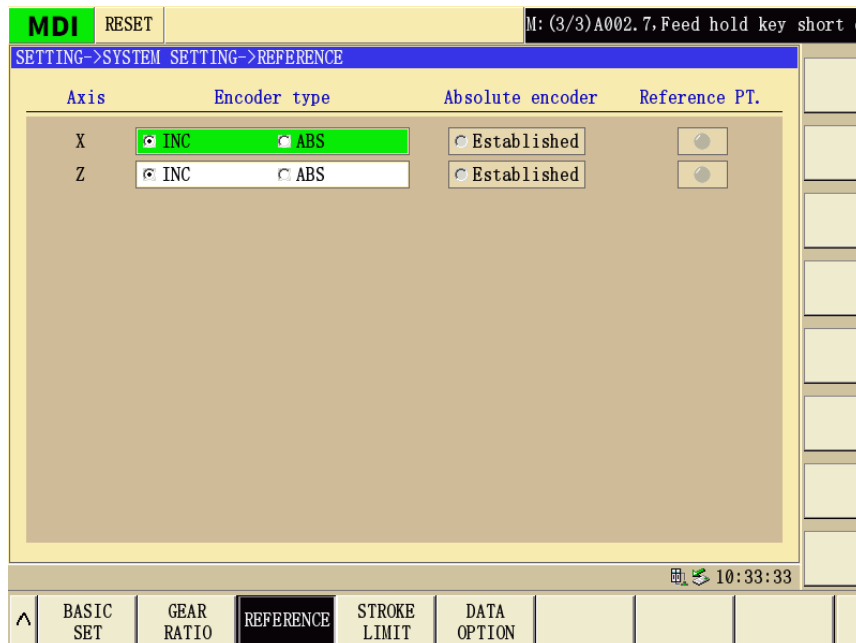


Fig. 1-62

Under the absolute encoder type, select "reference point establishment" to set the current position as the zero position, as shown in Fig. 1-63.



Fig. 1-63

1.4.5.2 Set travel limit

Press  →  to enter the setting page, as shown in Fig. 1-64.



Fig. 1-64

This page can display or set the boundary coordinate values of the two detection areas of stroke detection 1, and the current machine tool coordinate values.

With  soft keys, the machine coordinates of the current axis can be quickly input into the input box where the cursor is located.

The parameters involved in the modification of the reference point setting function are shown in the following table.

	Parameter Number		Note
Set reference point	1815	BIT4	Reference point establishment
		BIT5	Incremental and absolute
Set moving direction	1811	BIT2	The pulse output direction of each axis is reversed
Stroke limit	1320		Positive boundary of zone I
	1321		Negative boundary of zone I

1.4.6 One-key backup/recovery function

Press  →  to enter the setting page, as shown in Fig. 1-65.

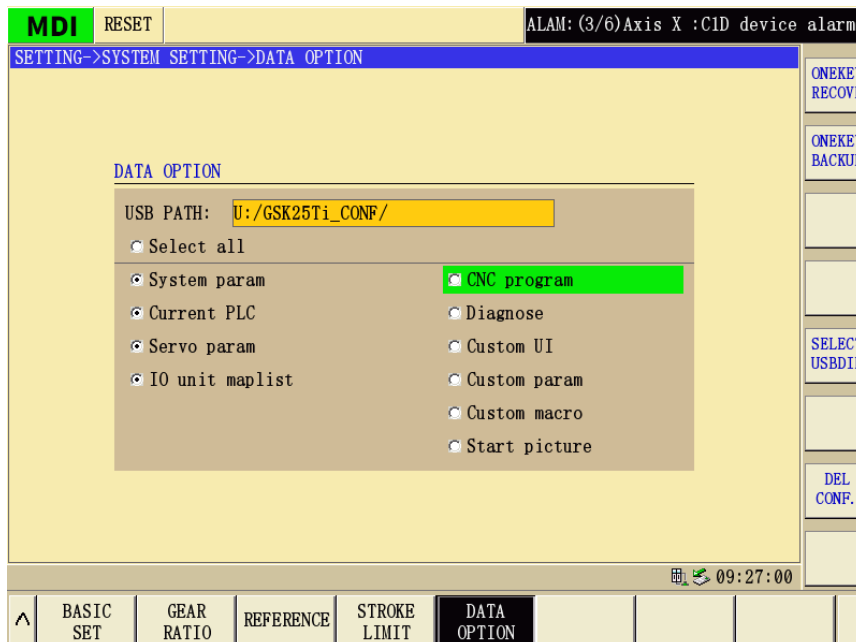
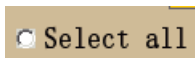


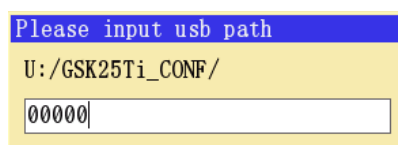
Fig. 1-65

The data operation page allows backing up files such as "System Parameters", "Servo Parameters", and "Machining Programs" to a USB flash drive, or restoring such files from a USB flash drive. After the system is powered on, the default "USB Drive Path" is the "U:/GSK25Ti_CONF/" directory(the directories of different CNC systems are different). Click the  soft key to reselect a new USB drive working path.

1.4.6.1 One-key backup

1、 First, use the direction keys to navigate, then click the  key on the editing keyboard to check the options you need to back up. If you need to back up all options, simply click the  option to check all items.

2 、 After inserting the USB flash drive, press the  soft key. A prompt dialog box



will pop up. Enter a folder name for saving the data, such as "00000", then click  to complete the backup process. The data backed up by One-Click Backup will be

written into the following folder (assuming the path selected when clicking "Select USB Drive Directory" remains "U:/GSK25Ti_CONF/").

Name	Path	Note
USB Flash Drive Backup Directory	U: /GSK25Ti_CONF/00000/	Among them, "00000" is the folder name set in the dialog box.

3、The backup locations of each option file are as follows:

Option File	Path	Note
Machining Program	U: /GSK25Ti_CONF/00000/CNCPROG	
Machine Tool Diagnosis	U: /GSK25Ti_CONF/00000/DIAGNOSE	
Customized Interface	U: /GSK25Ti_CONF/00000/UI	
Other Files	U: /GSK25Ti_CONF/00000	The fixed name of the system parameters is PARAM.PAR. PLC programs end with the .LD2 suffix. Servo parameters end with the .SPR suffix. The fixed name of the I/O mapping table is IOGLM.CFG. The fixed name of the custom parameters is CUSTOM.RGL. Custom macro variables end with the .EXT suffix. The fixed name of the startup picture is start.gif.

1.4.6.2 One-key recovery

Press the  soft key to restore the previously backed-up files from the USB flash drive.

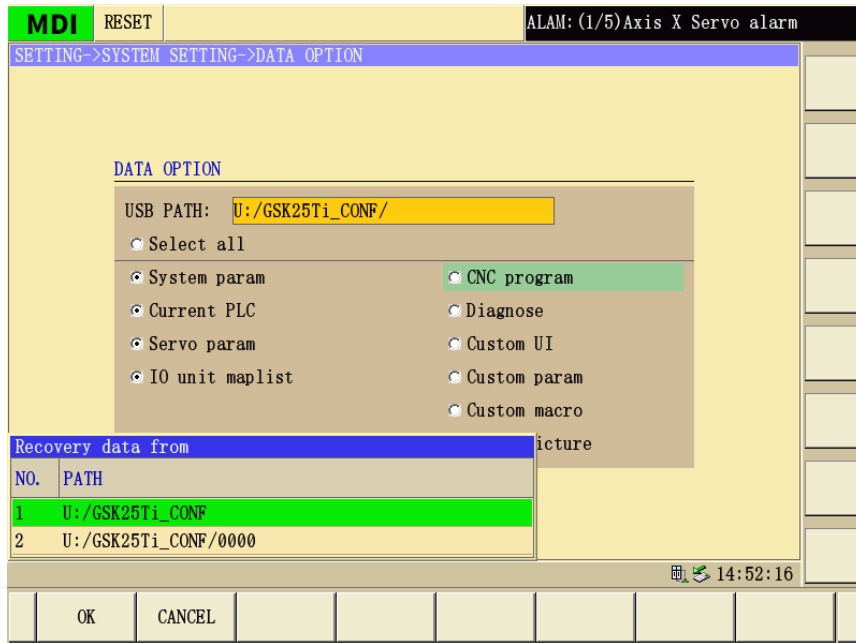


Fig. 1-66

After checking the options to be restored, press the  soft key to start restoring data from this directory. The process is as follows:

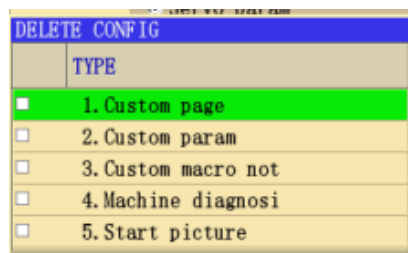
- 1、Click  to choose the path where files were previously backed up. The default path upon power-on is the "U:/GSK25Ti_CONF/" directory.

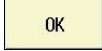
- 2、Move the direction keys and press the  key to check the file options that need to be restored.

- 3、Click the  key, select the previously backed-up folder name (as shown in Figure 1-66 as "U:/GSK25Ti_CONF"), and then click  to complete the restoration of the selected files.

1.4.6.3 Delete configuration

Press the  key, and the "DELETE CONFIG" dialog box will pop up, as shown in the figure below.

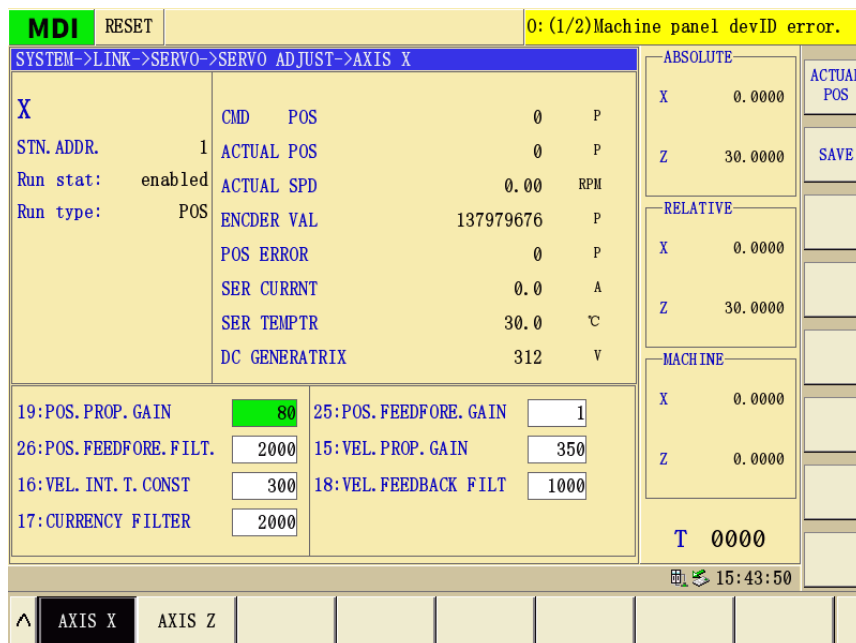


Within this dialog box, check the options that need to be deleted and click —the files corresponding to the selected options in the system will be deleted, and the change will take effect after the system is powered on again.

1.5 Servo debugging operation

1.5.1 Servo adjustment

Under the system page set, press  →  →  to enter the servo diagnosis page.



MDI		RESET	0: (1/2) Machine panel devID error.	
SYSTEM->LINK->SERVO->SERVO ADJUST->AXIS X				
X	CMD POS	0	P	ABSOLUTE
STN. ADDR.	1	ACTUAL POS	0	P
Run stat:	enabled	ACTUAL SPD	0.00	RPM
Run type:	POS	ENCODER VAL	137979676	P
		POS ERROR	0	P
		SER CURRNT	0.0	A
		SER TEMPTR	30.0	℃
		DC GENERATRIX	312	V
19: POS. PROP. GAIN	80	25: POS. FEEDFORE. GAIN	1	
26: POS. FEEDFORE. FILT.	2000	15: VEL. PROP. GAIN	350	
16: VEL. INT. T. CONST	300	18: VEL. FEEDBACK FILT	1000	
17: CURRENCY FILTER	2000			
				RELATIVE
				X 0.0000
				Z 30.0000
				MACHINE
				X 0.0000
				Z 0.0000
				T 0000
15:43:50				
^ AXIS X AXIS Z				

Fig. 1-67

The System Servo Adjustment Module provides the following functions:

Real-time monitoring of system control axis through servo communication feedback data, so that the operator can understand the current working status of servo drive units, motor and other equipment, including servo logic ID number, running status, running mode, command position, feedback position, motor encoder Monitoring of value, position deviation, servo current, servo temperature, motor temperature, DC bus voltage, etc. At the same time, there are some simple servo adjustment functions on this page. Such as simple gain adjustment of position and speed.

The description of each data display area on the servo diagnosis interface is as follows.

	Significance
X	The axis name of the currently selected axis
STN.ADDR.	The ID number of the slave connected to the axis
Run stat	Servo current operating status
Run type	Corresponding servo control method
CMD POS	The number of position pulses received by the servo from the system (displayed in position control mode)
CMD SPD	The command speed value received by the servo from the system (displayed in speed control mode)
ACTUAL POS	Position pulse number of servo feedback
ENCODER VAL	Encoder value of current motor position
POS ERROR	Deviation value between command position and feedback position
ACTUAL SPD	The actual speed of the motor
SER CURRNT	Servo current working current value
SER TEMPTR	Temperature measurement value inside the servo
MOTOR TEMP.	The temperature of the motor controlled by the servo
DC GENERATRIX	Servo current DC bus voltage value

Axis switching:

Press Axis X, Axis Z, Axis CS to switch between the X-axis, Z-axis, and CS axis to display the servo parameters.

Note: The servo parameters modified on this page are not saved and will be lost after the servo is powered off. If you need to save, please go to the servo parameter page and press the [Save] button to save.

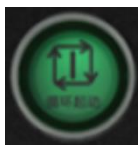
1.5.2 Roundness test

After the system is successfully connected to the GSK-Link bus communication of the servo, you can observe the roundness test chart during arc cutting through the servo debugging function on the system side, judge whether the current servo parameters are set to the most reasonable and optimized, and analyze each servo The dynamic response characteristics on the current loop/speed loop/position loop, and the roundness interpolation error range of the two axes. If you need to modify the parameters, you can press the SERVO ADJUST soft key under the servo debugging page to enter the parameter setting page. At this time, you can reset the speed loop proportional gain, position loop proportional gain and other parameters.

On the system page, press LINK → SERVO TUNE to enter the servo debugging function, and press the

**RADIUS
FEED**

soft key to enter the roundness test page. After setting the commands that need to be executed for the roundness test on the command setting page, switch to the **ANALY** page, press **SAMPLE** the soft key, and



then press the key in the MDI mode to execute the roundness test command. After the system is executed, the roundness diagram will be automatically generated.

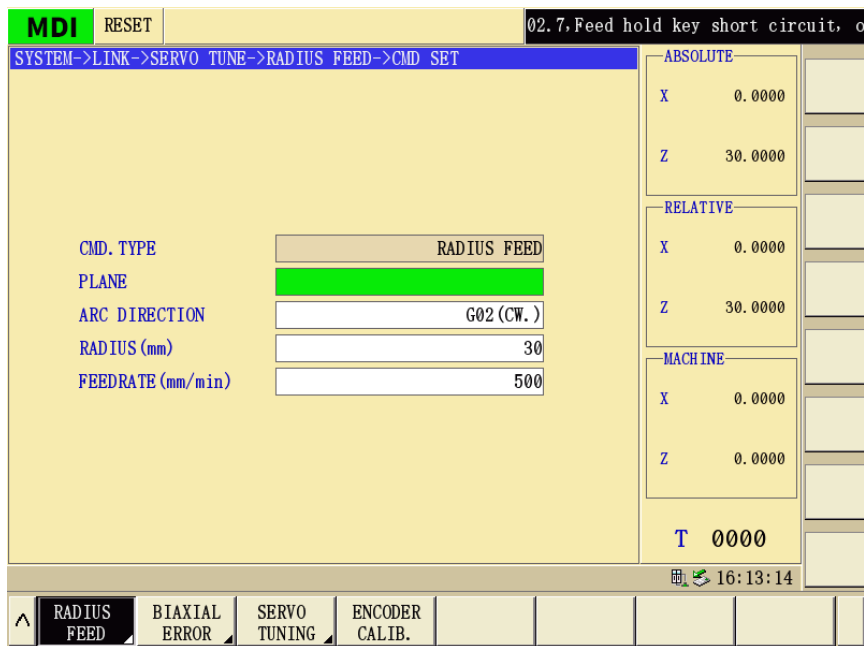


Fig. 1-68 Command setting page

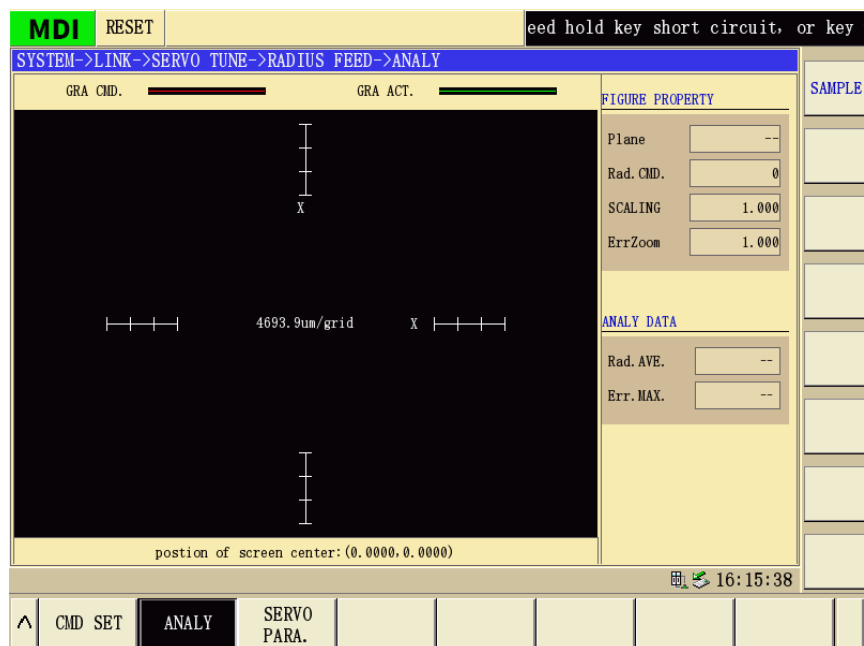


Fig. 1-69 Roundness diagram

1.5.3 Biaxial error

On the system page, press **LINK** → **SERVO TUNE** → **BIAXIAL ERROR** to enter the dual-axis error page,

as shown in Figure 1-70. Press the soft key after setting the reference axis and dual-axis synchronization

SAMPLE

equivalent, and then press the **STOP** **SAMPLE** soft key after running for a period of time. At this time, the two-axis synchronization error graph and following error information are automatically generated.

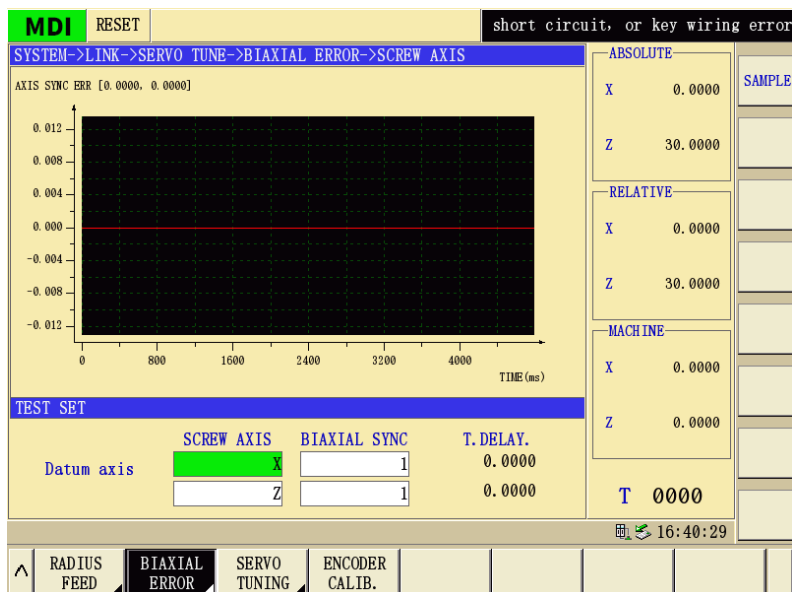


Fig. 1-70

1.5.4 Servo tuning

Press the **SYS** **系统** → **LINK** → **SERVO TUNE** → **SERVO TUNING** key to enter the tuning page, as shown in Figure 1-71.

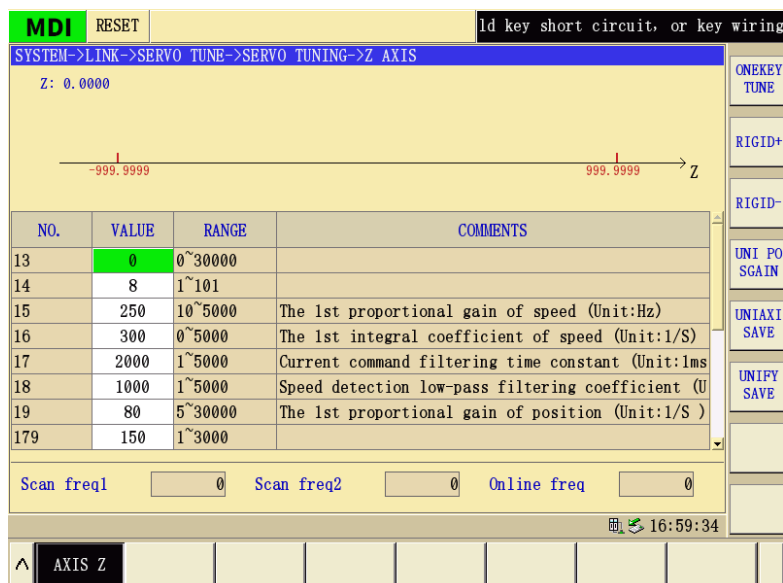


Fig. 1-71

1.5.5 Encoder Calibration

Press the  →  →  →  key to enter the encoder calibration page, as shown in Figure 1-72.

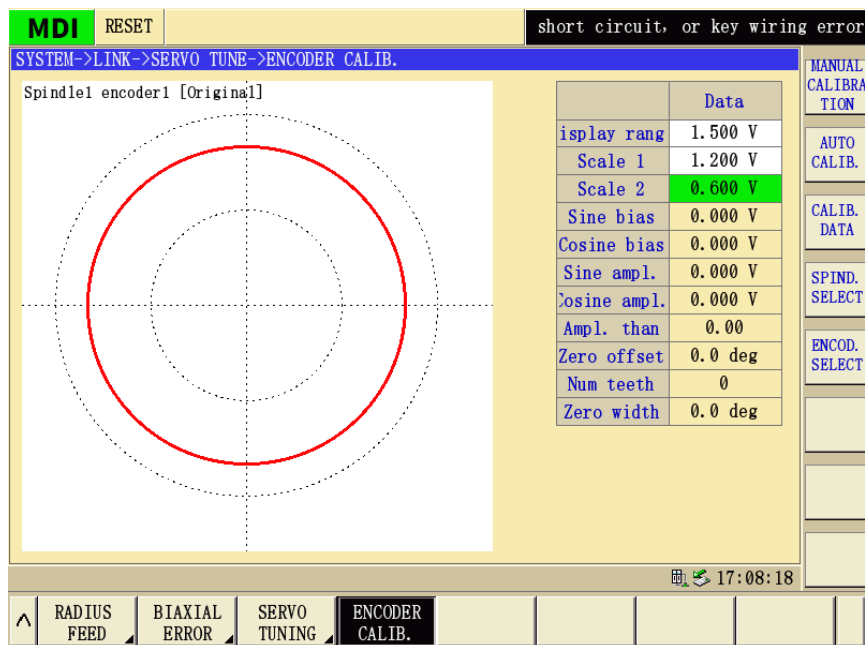


Fig. 1-72

1.6 Diagnostic information

1.6.1 System diagnosis

The system diagnosis includes editing keyboard diagnosis, hardware interface diagnosis, bus status diagnosis and communication data diagnosis.

1.6.1.1 Edit keyboard diagnostics

Press the  key to enter the information interface, and press the  →  soft key to enter the editing keyboard diagnosis page, as shown in Figure 1-73.

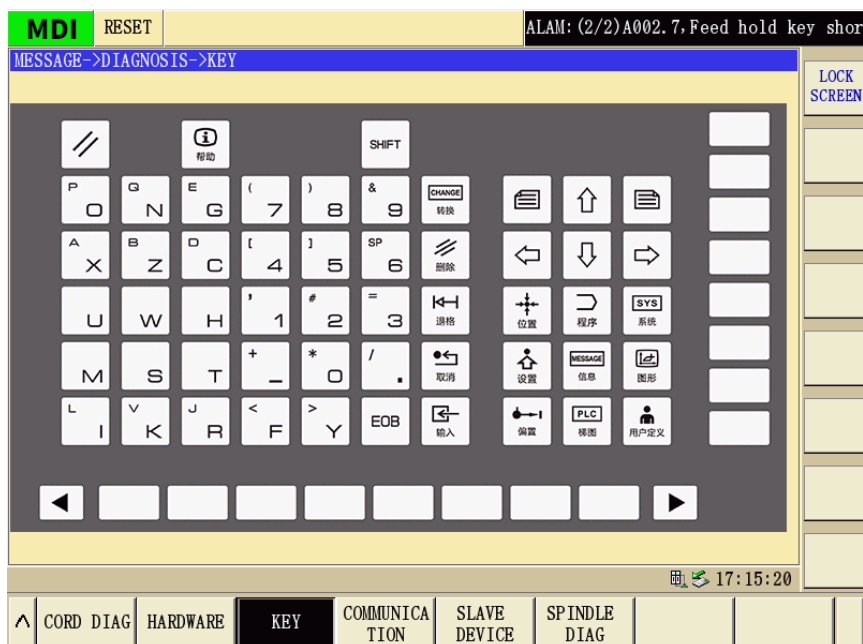


Fig. 1-73

The editing keyboard diagnostic page displays the system analog editing keyboard. When there is a key action, the corresponding key will be reversed, in order to prevent the diagnostic information of some keys (such as: arrow keys and page turning) from being generated. Corresponding function operation, you can lock the current screen through the **LOCK SCREEN** soft key.

1.6.1.2 Hardware interface diagnosis

Press the **MESSAGE** key to enter the information interface, and press the **CNC DIAGNOS** → **HARDWARE** soft key to enter the hardware interface diagnosis page, as shown in Figure 1-74.

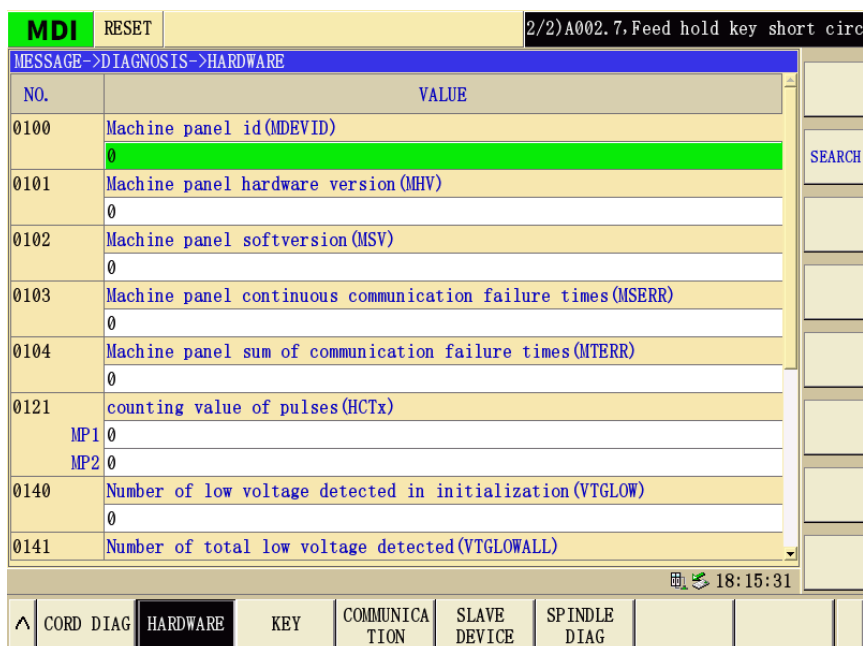


Fig. 1-74

The hardware interface diagnosis page displays the current CNC hardware information, hardware error information and MPG count data.

1.6.1.3 Bus status diagnosis

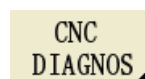
Press the  key to enter the information interface, and press the  →  soft key to enter the bus communication diagnosis page, as shown in Figure 1-75.

MDI		RESET		short circuit, or key wiring error				
MESSAGE->DIAGNOSIS->COMMUNICATION								
NO.		VALUE						
0400		FPGA version (VFPGA)						
		253						
0401		GLM version (VGLM)						
		1411						
0410		Connection state of Link (GLM)						
		1						
0411		Current initial step (STEP)						
		6						
0412		Number of servo slave devices (NUMSER)						
		3						
0413		Number of common slave devices (NUMCOM)						
		3						
0420		Receiving statuw of GDT transmission (GDTR)						
		0						
0421		Sending status of GDT transmission (GDTS)						
		0						
18:17:31								
^	CORD DIAG	HARDWARE	KEY	COMMUNICA TION	SLAVE DEVICE	SPINDLE DIAG		

Fig. 1-75

The bus status diagnosis page displays the status information of the equipment connected to the current CNC bus, including the number of slave stations on the bus, the communication stage and its communication error information.

1.6.1.4 Communication data diagnosis

Press the  key to enter the information interface, and press the  →  soft key to enter the communication data diagnosis page, as shown in Figure 1-76.

MDI

RESET

eed hold key short circuit, or key

MESSAGE->DIAGNOSIS->SLAVE DEVICE

NO.	VALUE
0600	MDT data field to servo(PMDT)
X	0.0
Z	0.0
0601	AT data field from servo(PAT)
X	0.34
Z	0.34
0606	MDT control field to servo(PMDTC)
X	H:c030
Z	H:c030
0607	AT state field from servo(PATS)
X	H:c010
Z	H:c010
0620	MDT data field to spindle(SMDT)
S	0
0621	AT data field from spindle(SAT)
S	0

18:19:51

CORD DIAG	HARDWARE	KEY	COMMUNICA TION	SLAVE DEVICE	SPINDLE DIAG		
-----------	----------	-----	-------------------	-----------------	-----------------	--	--

Fig. 1-76

The communication data diagnosis page displays the real-time data of the servo and remote I/O currently connected via the bus.

1.6.2 Bus connection diagnosis

LINK DIA

Under the diagnosis information page, press the soft key to enter the GSK-Link diagnosis page. The page is displayed as shown in Fig. 1-77.

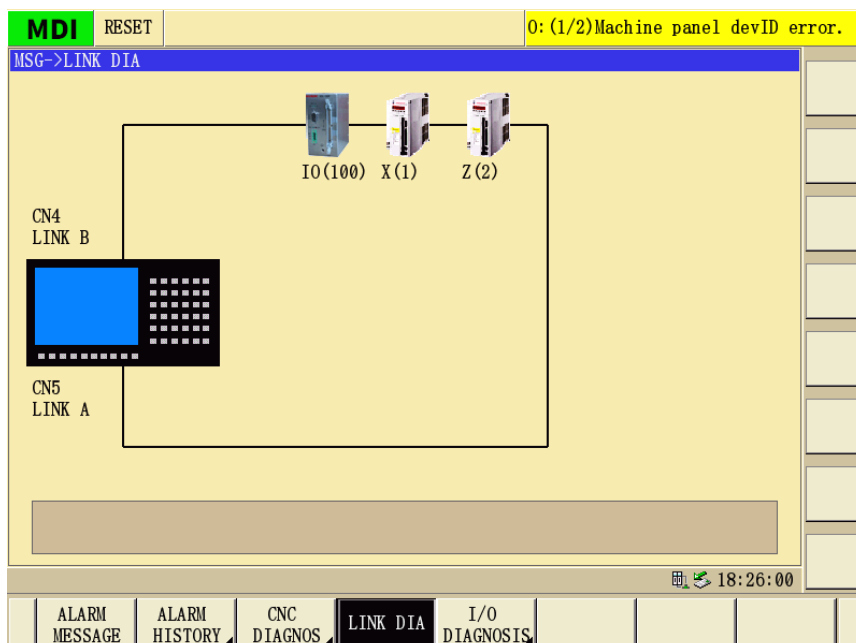


Fig. 1-77

When the GSK-Link function is enabled, this page is displayed. The GSK-Link diagnosis page mainly

diagnoses the current communication situation. When a broken ring occurs, the location of the broken ring and the cause of the fault are displayed on this page.

1.6.3 I/O diagnosis

In the diagnosis information interface, press **I/O DIAGNOSIS** the soft key to enter the I/O diagnosis page. The page is displayed as shown in Fig. 1-78.

MDI

RESET

INFO: (1/2)Machine panel devID error

MESSAGE->I/O DIAGNOS->LOCAL IO

ADDRESS		DISCRIPTION	NO.		DISCRIPTION
X0.0		spare	Y31.0		
X0.1		spare	Y31.1		
X0.2		Manual tool setting signal	Y31.2		
X0.3		spare	Y31.3		
X0.4		G31 jump signal	Y31.4		
X0.5		ESP	Y31.5		
X0.6		G36 jump signal	Y31.6		
X0.7		G37 jump signal	Y31.7		
X36.0					
X36.1					
X36.2					
X36.3					
X36.4					
X36.5					
X36.6					
X36.7					

18:28:38

LOCAL IO

I/O UNIT1

Fig. 1-78

The I/O diagnosis display page includes information such as local I/O and I/O unit. Press the corresponding soft key to switch to the corresponding page.

Local I/O Diagnosis

Enter the diagnosis information page, press the **I/O DIAGNOSIS** soft key to enter the I/O diagnosis page, and then press **LOCAL IO** the soft key to enter the local I/O diagnosis page, as shown in Fig. 1-78.

The local I/O display page is mainly for diagnosing the hardware I/O information of the built-in IO port on the CNC system and whether there is any fault on the hardware of the I/O unit. You can use the ,



key or search to view the corresponding information on the page.

I/O Unit Diagnosis

In the I/O diagnosis page, press the **I/O UNIT1** soft key to enter the I/O unit diagnosis page, as shown in Fig. 1-79.

MDI

RESET

short circuit, or key wiring error

MESSAGE->I/O DIAGNOS->I/O UNIT1

NO.	DISCRIPTION	NO.	DISCRIPTION	SEARCH
X100.0	Opening detection signal of	Y100.0	cooling signal output	
X100.1	Protection door closing detection	Y100.1	lubricating signal output	
X100.2	chuck input signal	Y100.2	Hydraulic output signal	
X100.3	spare	Y100.3	spindle CW signal	
X100.4	tailstock control signal	Y100.4	spindle CCW signal	
X100.5	spare	Y100.5	spare	
X100.6	pressure detection signal	Y100.6	Spindle brake output signal	
X100.7	Sensor E(Lio Shing)	Y100.7	spare	
X101.0	TOOL-Loosen in place/Sensor	Y101.0	1st gear of spindle output signal	
X101.1	TOOL-Lock in place	Y101.1	2nd gear of spindle output signal	
X101.2		Y101.2	3rd gear of spindle output signal	
X101.3	Low lubricating oil level	Y101.3	4th gear of spindle output signal	
X101.4	spare	Y101.4	external chuck clam output/signal	
X101.5	1st shift in place	Y101.5	external chuck release output	
X101.6	2nd shift in place	Y101.6	tool change positive output	
X101.7	T01/Sensor A(Lio Shing)	Y101.7	tool change negative output	

18:34:35

LOCAL IO

I/O UNIT1

Fig. 1-79

This page is mainly used to diagnose the hardware I/O information of the I/O unit and whether there is any fault on the I/O unit hardware. For example, when the PLC diagnostic I/O output signal is valid, but the output on the machine tool is invalid, you can perform diagnosis on this page To see if the hardware of the

I/O unit is faulty. In the page, you can use



key or search to view the corresponding information.

1.7 Use of GSKUI software

1.7.1 Software overview

The GSKUI software is used for previewing and modifying configuration files related to custom pages. It supports the layout of custom interfaces and machine tool diagnostic pages, the reading and writing of custom macro variables and custom parameter files, and the management of startup images. The display layout of the software is as shown in Figure 1-80.

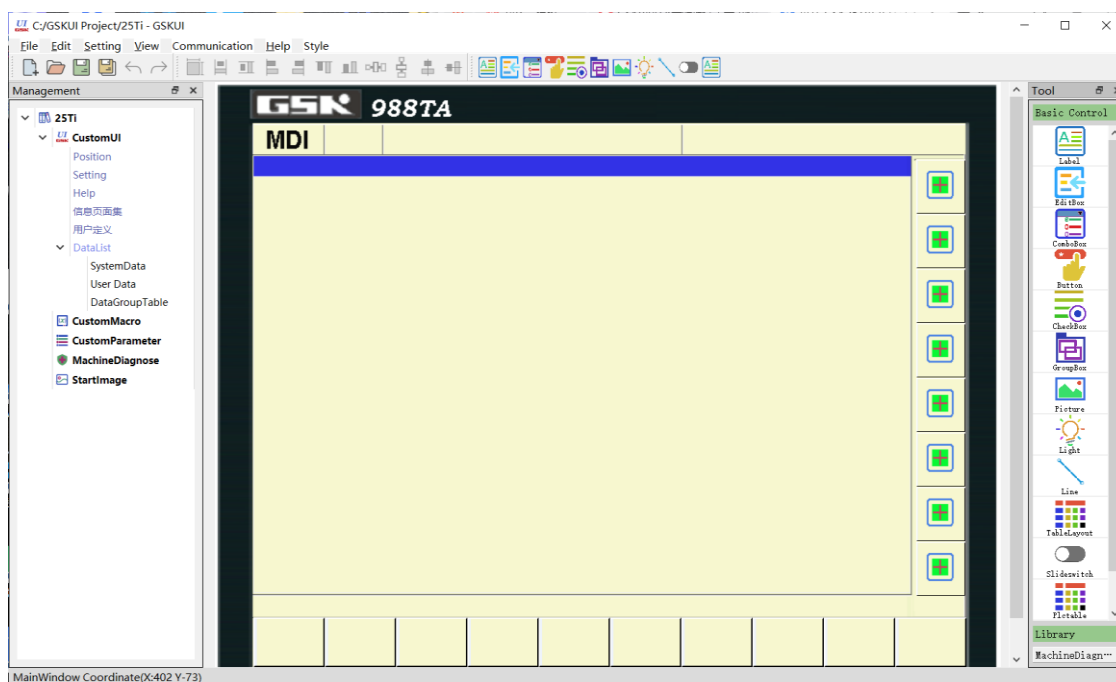


Fig. 1-80 GSKUI software interface

Custom Interface: Users can add customized pages under the system's "Setting" ([Setting](#)), "Position" ([Position](#)), and "Help" ([Help](#)) page sets.

Custom Macro Variables: Macro variables that need to be used can be integrated into a single page for easy viewing and modification.

Custom Parameters: Users can integrate required system parameters or PLC parameters into a single set by writing configuration files, facilitating parameter configuration and debugging.

Machine Tool Diagnosis: Users can add a "MachineDiagnose" ( **MachineDiagnose**) page under the information page set, which can use indicators to simply display the status of specific PLC addresses.

Startup Image: Manage a startup image, which can be transferred to the system and displayed during system power-on initialization.

After installing this software, click the "User Manual" ( User Manual F1) menu item under the "Help" ([Help](#)) menu to obtain more detailed operation instructions.

Note: To use the custom interface function, parameter No. 8132.2 must be enabled first.

1.7.2 Usage method

(1) Double-click the application icon to open the GSKUI software.

(2) Click "File -> New" ( New Ctrl+N) to pop up the New Project dialog box.

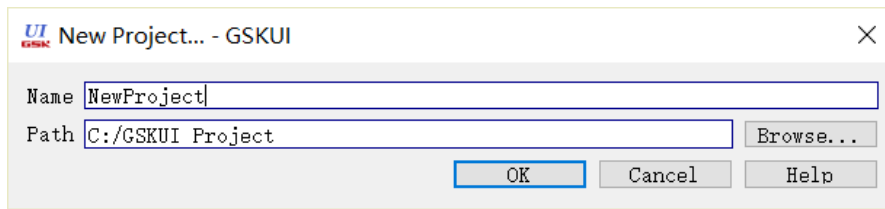


Fig. 1-81

(3) Enter the project name and select the storage path. As shown in Figure 1-81, a new project "NewProject" is created under the storage path "C:/GSKUI Project" directory, and a "NewProject" project folder will be created in the storage path at this time.

(4) Click "OK" to complete the project creation. The main interface of the new project is shown in Figure 1-82.

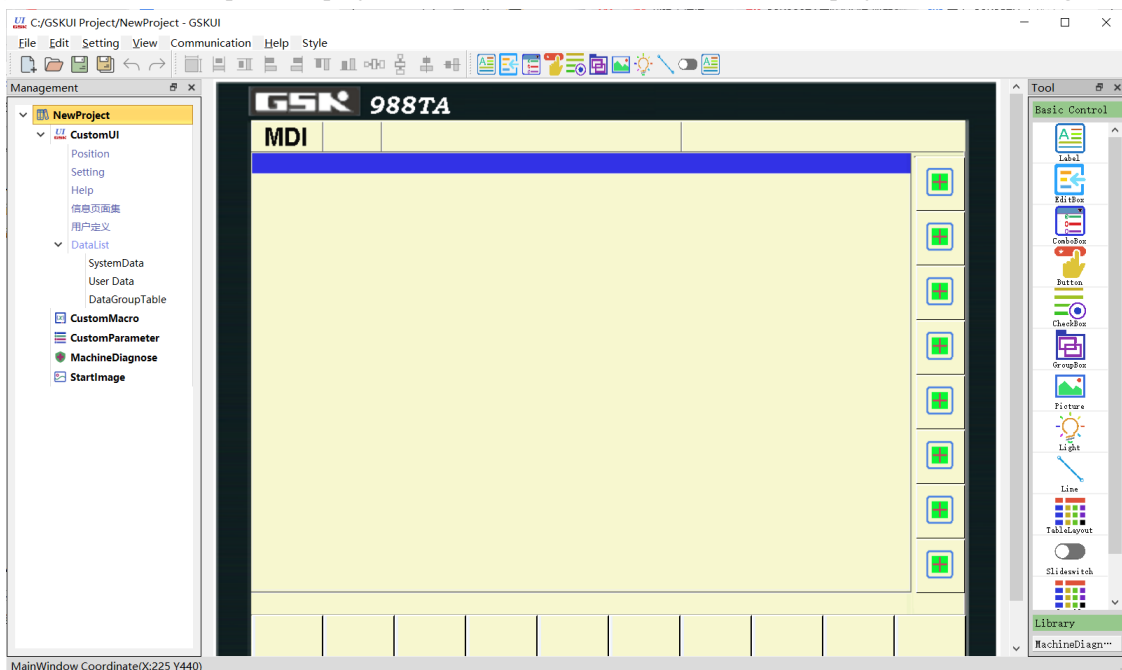


Fig. 1-82

(5) Under "CustomUI" (CustomUI), there are three nodes: "Position" (Position), "Setting" (Setting) and "Help" (Help), as shown in Figure 1-83. Right-click on these three nodes and select "New Function" to create a "Function" under the current node, and there is a page under the function.

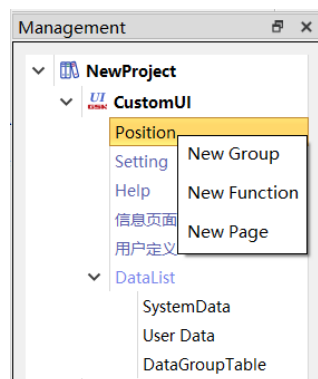


Fig. 1-83

- (6) Double-click the "Page" ( Page1) node to edit the content of that node's page. Right-clicking under other nodes will display different menu functions. For specific functions, you can double-click the "Help -> User Manual" menu, which contains more detailed explanations.
- (7) Use the left mouse button to click and hold a control icon in the toolbar or the "Basic Control" () panel on the right, drag it to the editing area of the page, and then release the left mouse button. At this point, a control will be generated at the position where the mouse is released. As shown in the figure below, a drop-down box is generated in the editing area of "Page 1".

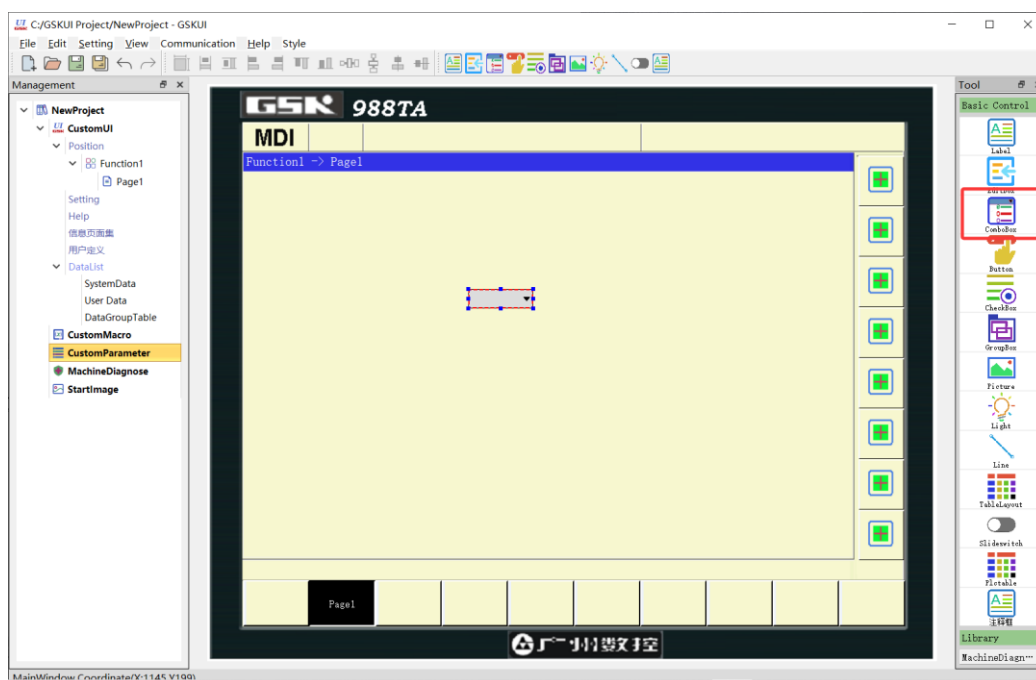


Fig. 1-84

- (8) After selecting a control by dragging the mouse to draw a selection box around it, you can reposition the control using the left mouse button or the arrow keys.
- (9) Double-click a control in the editing area to set its properties, such as operation address, displayed text, etc., as shown in Figure 1-85.

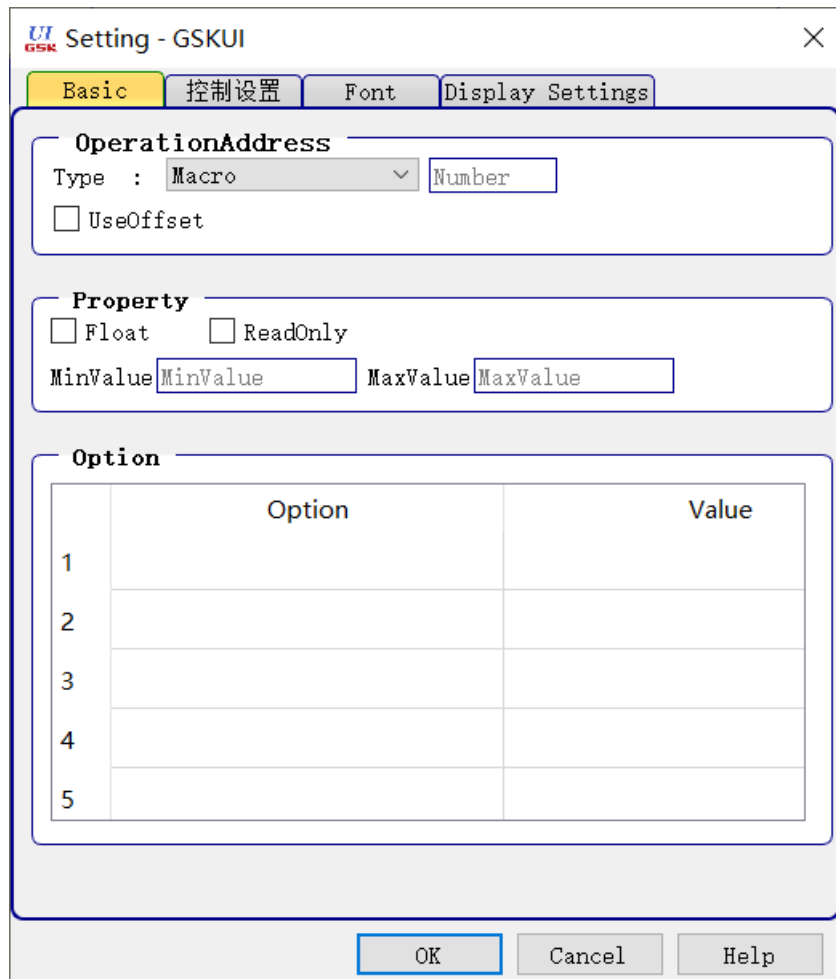


Fig. 1-85

- (10) After setting the properties, click "OK" to complete the settings; click "Cancel" to invalidate the property settings modified this time.
- (11) Repeat the above operations to add and edit other controls or pages.
- (12) After editing, the local project on the computer can be transferred to the system via network transmission or USB flash drive transmission. Network transmission is used here as an example.
- (13) Ensure that the network connection between the computer and the system is normal.
- (14) Click "Communication -> NetSetting", as shown in Figure 1-86.

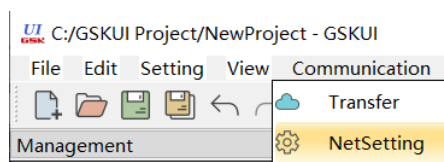


Fig. 1-86

- (15) The IP Settings dialog box will pop up. Enter the system's IP address here (the system's IP setting

information can be found on the system's "  ->  ->  " page).

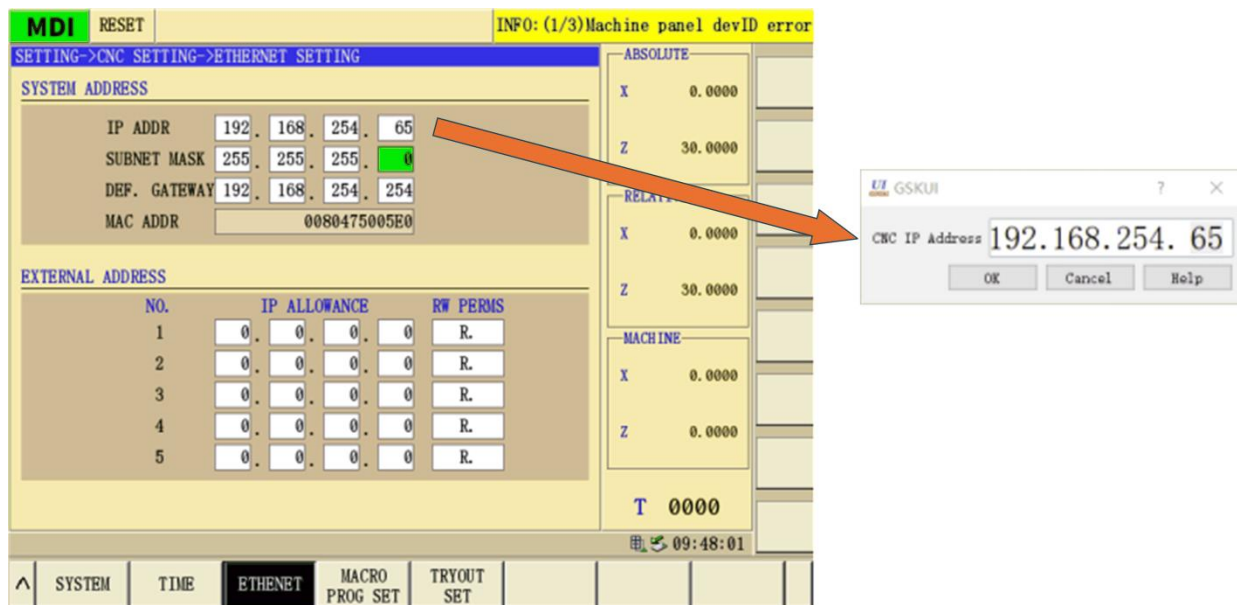


Fig. 1-87

(16) Click "OK" to complete the IP setting.

(17) Then click "Transfer".

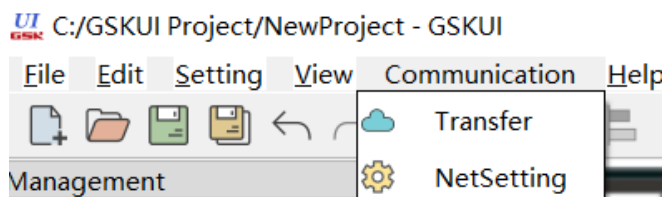


Fig. 1-88

(18) The "Transfer" dialog box will pop up. In the "Local" (left side of the figure below), select the type of file to be downloaded to the system, then click the arrow to start transferring the corresponding local files to the CNC.

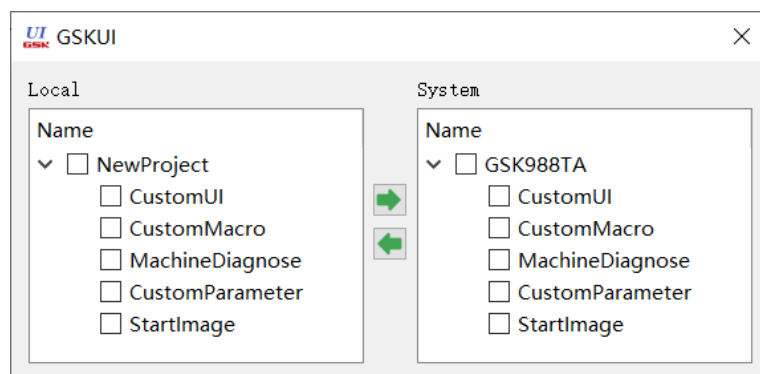


Fig. 1-89

(19) The progress bar of the file transfer process is shown in Figure 1-90.

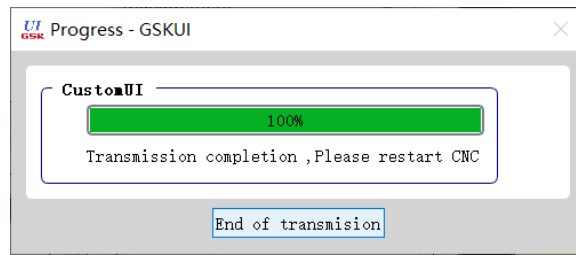


Fig. 1-90

- (20) After the transfer is completed, click the "End of transmission" button to finish the current transfer process.
- (21) After the files are downloaded to the system, the system needs to be powered off and on again for them to take effect.
- (22) A complete editing process is concluded.
- (23) The process of transferring files from the CNC to the local computer is basically the same. Once the files are transferred to the local computer, they will automatically overwrite the original project. Therefore, when transferring project files from the CNC to the local computer, it is recommended to create a new empty project first before performing the transfer.

1.7.3 Explanation of project directory

1.7.3.1 Explanation of project path

Path Name	Description	Remarks
{proj }/	Indicates the directory where the opened project file is located.	
{xxx}	Indicates the name of the opened project file.	Entered by the user when creating a project.

1.7.3.2 Directory structure explanation

File Name	Description	Remarks
Gskui.upj	The overall project file.	Fixed file name: Gskui.upj Located in the {proj}/UI directory.
gskui.cfg	Page set file.	Located in the {proj}/UI directory.
Group{yyy}_Page{zzz}.uic	Individual page display configuration file.	1 <= yyy < 8, 1 <= zzz < 6 Located in the {proj}/UI directory.
res/	Resource file directory.	Located in the {proj}/UI directory.
datalist.cfg	Data table file.	Located in the {proj}/UI directory.
*.ext	Custom macro variable file.	Located in the {proj}/ directory.
CUSTOM.RGL	Custom parameter file.	Located in the {proj}/ directory.
Machine tool diagnosis file(*.ini *.gif)	Machine tool diagnosis related files.	Located in the {proj}/Diagnose directory.
start.gif	Startup image.	Located in the {proj}/ directory.

1.7.4 File transfer

After editing the GSKUI project on the PC, there are two methods to transfer the project files between the PC and the CNC system.

1.7.4.1 USB flash drive transfer

The import and export methods are described for CNC system version V2.22c and later. The location of the import and export pages may vary slightly in other system versions, but the operation methods are largely similar.

Copy the entire edited custom interface project folder to the USB flash drive. Assuming the project folder is named "PROJECT", you can copy this project folder to the root directory of the USB flash drive (or place it in other directories).

Configuration file import

- (1) Navigate to the "  ->  ->  " page of the system.
- (2) With Level 2 permissions, insert the USB flash drive, and check the items to be imported (such as  on the Data Operation page.
- (3) Click  - a dialog box will pop up as shown below. Select the file recovery path (e.g., "U:/PROJECT") as illustrated in Figure 1-91, indicating that files need to be restored from the "U:/PROJECT" directory.

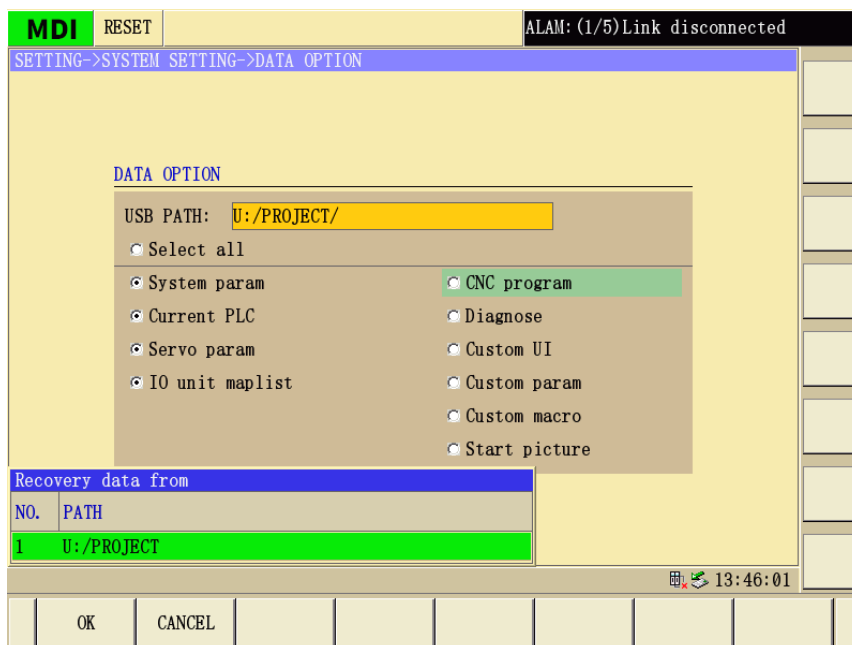


Fig. 1-91

- (4) Click "OK" to complete the import of the custom interface files.

Configuration file export

- (1) Navigate to the "  ->  ->  " page of the system.

- (2) With Level 2 permissions, insert the USB flash drive, and check the items to be exported (such as "Custom UI") on the Data Operation page.

- (3) Click **SELECT USBDIR**, then choose the location to store the project folder. We can select "U:", which is the root directory of the USB flash drive, as illustrated in Figure 1-92.

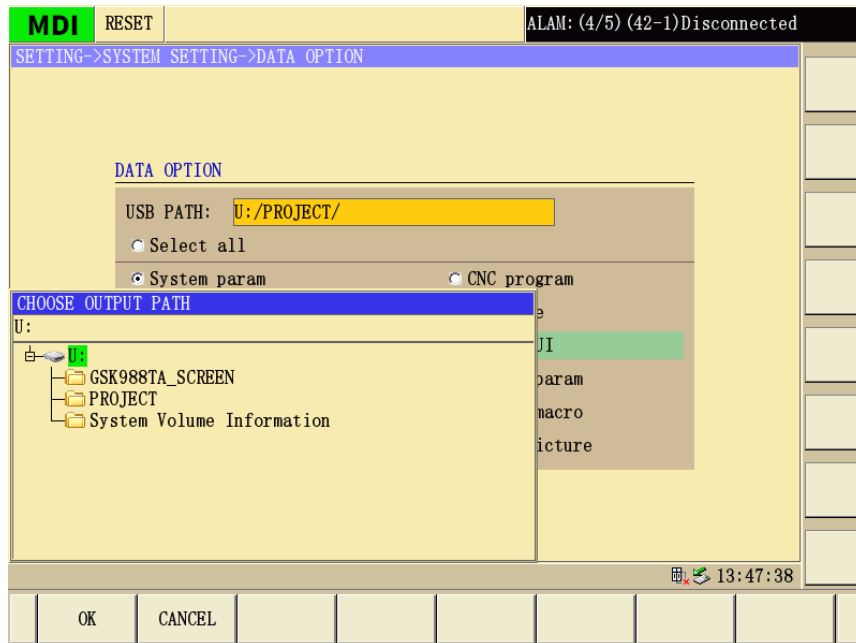


Fig. 1-92

- (4) Click **ONEKEY BACKUP**, then set the folder name (e.g., PROJECT), as illustrated in Figure 1-93.

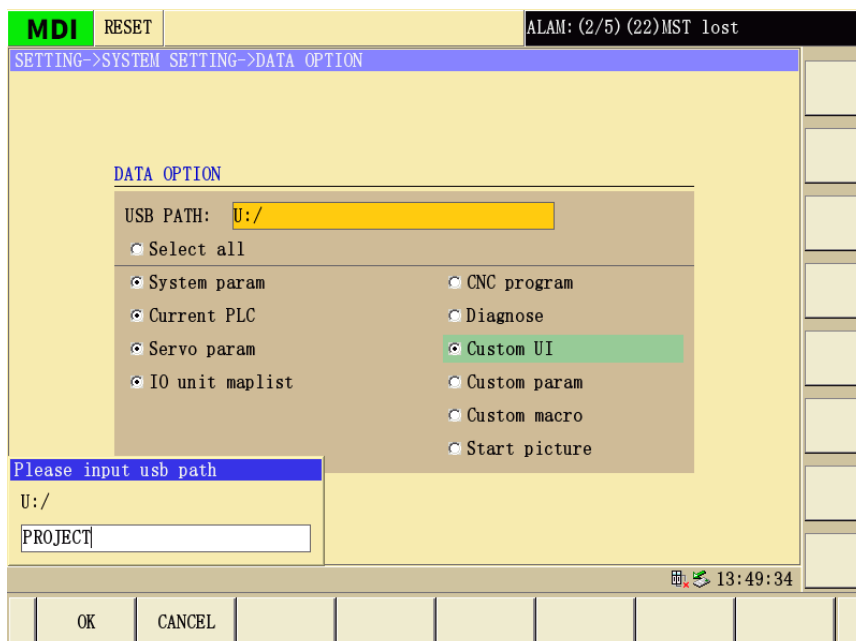


Fig. 1-93

- (5) Click OK to complete the export of the custom interface files. After export, "PROJECT" will be the

project folder with the path "U:/PROJECT", and the custom interface files will be placed in the UI directory under the PROJECT folder.

- (6) If you need to edit the project, copy the entire PROJECT folder to the computer, open it with GSKUI software, and select the "PROJECT" folder to open the project.

1.7.4.2 Network transfer

Communication settings

- (1) Whether uploading or downloading files, first ensure the network is unobstructed and the set system IP address is correct.

- (2) The current IP address of the system can be found under the " ->  -> 

- (3) Click "Communication -> NetSetting" in the software, and enter the system's IP address.

Transfer to CNC

Software versions later than V2.0.0 on the PC support the file transfer function, which enables convenient uploading and downloading of project files. The operation steps are as follows:

- (1) Complete the editing of project files in the software on the PC.
- (2) Click the "Communication ->  NetSetting " menu item, and set the system's IP address (the system IP

can be viewed on the " ->  -> 

- (3) Click the "Communication ->  Transfer" menu item, and a "File Transfer" dialog box will pop up. Check the items that need to be downloaded to the CNC.

- (4) Then click the arrow  to start transferring local files to the CNC. After the transfer is completed, restart the system to make the settings take effect.

Transfer from CNC

- (1) Click the "Communication ->  NetSetting " menu item, and set the system's IP address (the system IP

can be viewed on the " ->  -> 

- (2) Click the "Communication ->  Transfer" menu item, and a "File Transfer" dialog box will pop up. Check the items that need to be uploaded from the CNC to the local.

- (3) Click the arrow  to start transferring files from the CNC to the local.
- (4) After the files are transferred to the local, they will automatically overwrite the project being edited.

Therefore, when transferring project files from the CNC to the local, it is recommended to create a new empty project before transferring the files.

Chapter 2 Machine Debugging and Function

2.1 GSK-Link bus connection

The system is connected to the feed servo drive unit, spindle drive unit, and I/O unit via the GSK-Link bus. Connect the components as shown in Fig. 2-1.

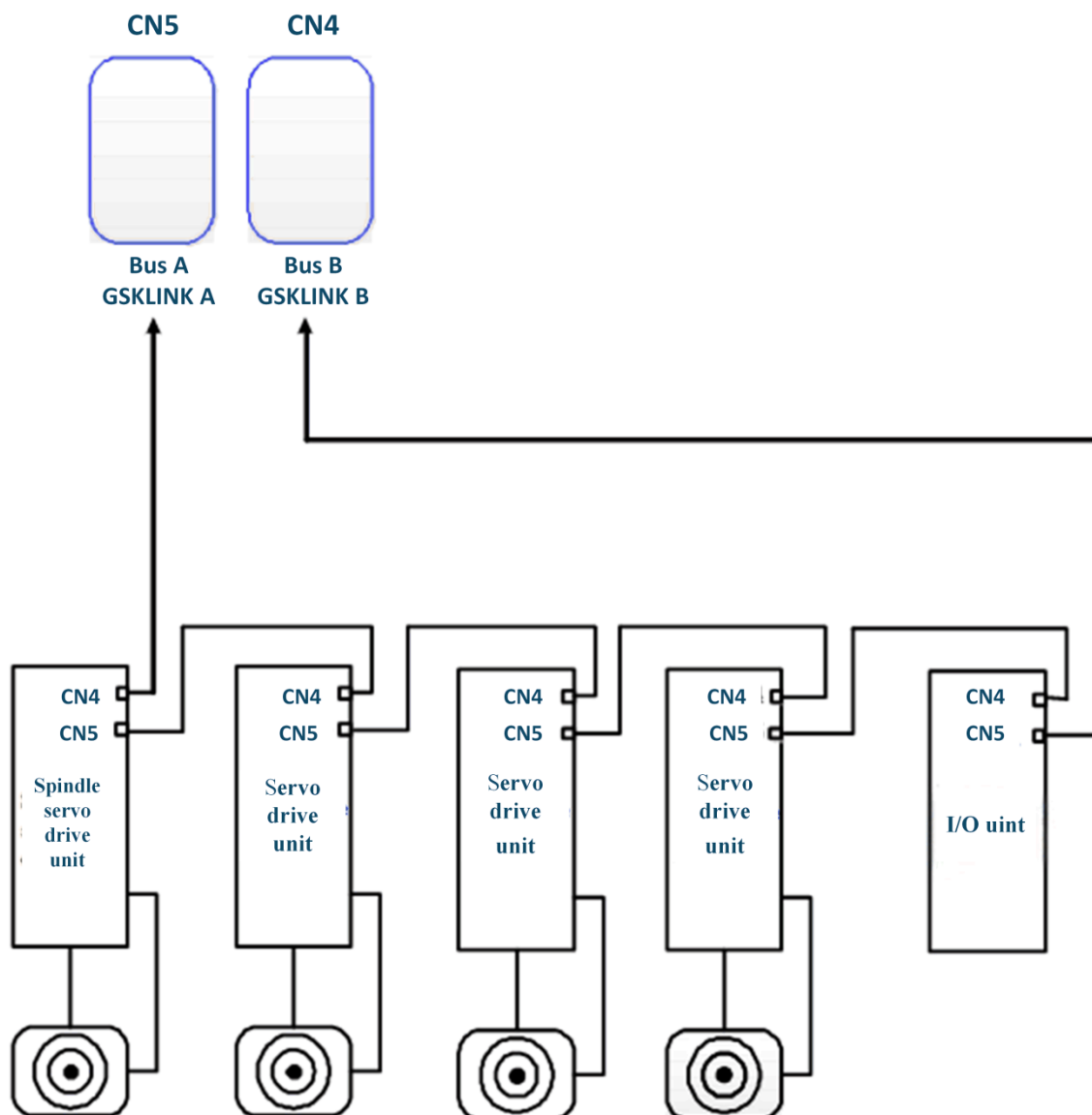


Fig. 2-1

Relevant parameters must be set to ensure the equipment operates normally. The related parameters are shown in Table 2-1.

Table 2-1

Relevant Parameters					
	Model	Parameter Number	Bit	Parameter Signification	Note
CNC	parameter	9000	#0	Whether the system GSK-Link communication function is valid 0: invalid 1: valid	
		1010		Set the maximum number of axes that can be directly controlled by the CNC (0 to total number of controlled axes). The remaining axes are controlled by the PLC.	
		1023		Logical ID number of each axis	The set axis number corresponds to the setting value of each axis drive unit
		3050		Number of I/Os controlled by the system	
		3051		Logical ID number of system control I/O unit 1	
		3052		Logical ID number of system control I/O unit 2	
		3053		Logical ID number of system control I/O unit 3	
		3054		The logical ID number of the system control I/O unit 4	
		3710		Set the number of spindles controlled by the CNC	
		3717		Amplifier number of each spindle	When it is set to -34 to -1, it means that the spindle is not connected by bus and is a normal analog spindle. When it is set to 1~99, it is a bus spindle, and the setting value must be consistent with the corresponding spindle drive unit.
		8410		Total auxiliary axes; 0 means the function is disabled.	
Servo	GR2000 series	PA156		GSK-Link communication servo slave number	Correspond to the logic ID number of each axis of CNC parameter 1023
		PA4		Control mode selection: 21 is the bus mode	
	Model	Parameter Number	Bit	Parameter Signification	Note
	GR3000 series	PA156		GSK-Link communication servo slave number	Correspond to each spindle amplifier number of CNC parameter 3717
		PA4		Control mode selection: 21 is the bus mode	

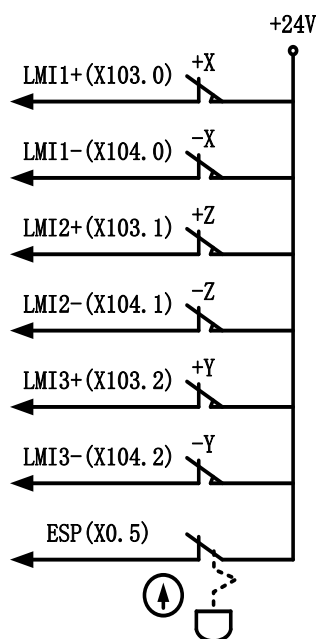
Relevant Parameters					
	Model	Parameter Number	Bit	Parameter Signification	Note
I/O Unit	IOL				The communication parameters are set by the system side. For the specific setting method, refer to 1.1.3 of "Second debugging"

Note 1: The slave number corresponding to GSK-Link communication cannot be repeated, otherwise the GSK-Link communication between the servo and the system cannot be established.

Note 2: The slave number in the servo needs to be manually modified by the user on the drive unit before connecting the servo to GSK-Link (it will take effect after re-power on).

2.2 Emergency stop and hard limit

The system has a software limit function, but for safety reasons, it is recommended to take hardware limit measures at the same time, install the travel limit switch in the positive and negative directions of each axis, and the connection is shown in Fig. 2-2.



Emergency stop

Fig. 2-2

In the "manual" or "MPG" mode, slowly move each axis to verify the effectiveness of the overtravel limit switch, the correctness of the alarm display, and the effectiveness of the overtravel release button. (Move the overtravel axis in the opposite direction, can release the alarm).

Table 2-2

Relevant Parameters				
	Parameter Number	Bit	Parameter Signification	Note
CNC Parameters	3003	#7	ESP external emergency stop alarm signal (X0.5) 0: Emergency stop alarm when the input signal is 0 (low level) 1: Emergency stop alarm when the input signal is 1 (high level)	These two parameters must be set to the same
Standard PLC Parameters	K0010	#7	External emergency stop alarm signal (X0.5) 0: Emergency stop alarm when it is low level 1: Emergency stop alarm when it is high level	
CNC Parameters	3004	#5	Overtravel limit signal 0: Check 1: Do not check	
Standard PLC Parameters	K0010	#2	Overtravel signal input level selection 0: Alarm at high level 1: Alarm at low level	

2.3 Basic axis parameter setting

(1) Axis name

The basic control axis of the system is 2 axes, the expansion control axis number is 6 axes (including Cs axis); the basic linkage control axis number is 3 axes.

Table 2-3

Relevant Parameters		
Parameter	Parameter Signification	Note
8130	The total number of control axes of the system	
1010	CNC control axis	This parameter value cannot be greater than parameter No. 8130
1020	Programming name of each axis	The name of each axis cannot be the same
1022	Attributes of Each Axis in the Basic Coordinate System	
1023	Servo axis number of each axis	The set axis number corresponds to the setting value of each axis drive unit

(2) Axis unit

After setting the above parameters, the user can set the equivalent of the system and the input/output unit of each axis according to the needs of use. You can also set the minimum input and output increments of the axis according to the requirements of machining accuracy.

Table 2-4

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
0000	#2	Input unit 0: Metric system 1: Imperial system
1001	#0	The minimum movement unit of the linear axis is 0: Metric system (Metric machine tool) 1: Imperial system (Imperial machine tool)
1004	#1	Set the minimum input unit and minimum command increment 0: 0.001mm 1: 0.0001mm
1006	#3	Set the movement amount of each axis as 0: Radius designation 1: Diameter designation

(3) Type of axis

During processing, the axis used can be set as a linear axis or a rotary axis according to the processing needs.

Table 2-5

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
1004	#6, #7	Set the minimum command increment of the rotary axis and the multiple of the ISC parameter 00: ×1 times . 01: ×10 times. 10: ×100 times
1006	#1, #0	00: Linear axis 01: Rotary axis (Type A) 11: Rotary axis (Type B)
1008	#0	Set whether the cycle display function of the rotary axis is valid 0: invalid 1: valid
1008	#1	Set the axis rotation direction when absolute command 0: The direction of rotation closer to the target 1: The direction specified by the command value sign
1260		The amount of movement per revolution of each axis when rotating the axis

2.4 Gear ratio calculation and setting

The most direct meaning of the electronic gear ratio is to make the moving distance of the machine tool carriage consistent with the program command value (the moving distance of the machine coordinate). If the axis is radius programming (radius/diameter programming setting: parameter 1006#3), the actual axis movement distance on the machine tool is equal to the machine coordinate movement distance displayed on the system; if the axis is diameter programming, the actual axis movement distance on the machine tool is Two times should be equal to the moving distance of the machine coordinate displayed on the system.

The gear ratio setting includes CNC gear ratio and servo drive unit gear ratio setting, which should be paid attention to in actual application.

Table 2-6

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
1001	#0	Minimum movement unit for linear axes 0: Metric system 1: Imperial system
1004	#1	Minimum input unit and minimum instruction increment 0: ISB system 1: ISC system
1006	#1、#0	Set linear axis or rotary axis 00: Linear axis 01: Rotary axis (A type) 11: Rotary axis (B type)
1006	#3	Set the movement amount of each axis as 0: Radius designation 1: Diameter designation
1816		Detection multiplication ratio of each axis
1820		Command multiplication ratio of each axis

Table 2-7

Incremental System						
	Input	Straight/ Radius Programming	IS—B		IS—C	
			Minimum Setting Unit	Minimum Movement Unit	Minimum Setting Unit	Minimum Movement Unit
Metric machine	Metric	Diameter	0.001mm	0.0005mm	0.0001mm	0.00005mm
		Radius	0.001mm	0.001mm	0.0001mm	0.0001mm
	Imperial	Diameter	0.0001 inch	0.0005mm	0.00001 inch	0.00005mm
		Radius	0.0001 inch	0.001mm	0.00001 inch	0.0001mm

Incremental System						
	Input	Straight/ Radius Programming	IS—B		IS—C	
			Minimum Setting Unit	Minimum Movement Unit	Minimum Setting Unit	Minimum Movement Unit
Imperial machine	Metric	Diameter	0.001mm	0.00005 inch	0.0001mm	0.000005 inch
		Radius	0.001mm	0.0001 inch	0.0001mm	0.00001 inch
	Imperial	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

2.4.1 Gear ratio calculation

Gear ratio calculation formula:

$$\text{Servo electronic gear ratio} = \text{Minimum movement unit} \times \frac{\text{Pulses per revolution of pulse encoder}}{\text{Lead}} \times \frac{Z_M}{Z_D} \times \frac{\text{CMR}}{\text{DMR}}$$

Minimum movement unit:

The minimum command unit transmitted from the CNC to the machine tool, which is also the minimum increment of the tool movement on the machine tool, also known as the minimum moving unit.

Pulses per revolution of pulse encoder:

= the number of encoder lines (absolute encoder used by the feed motor);

= 4 × the number of encoder lines (incremental encoder used by the feed motor).

Z_M : The number of teeth of the screw end gear.

Z_D : The number of teeth of the gear at the motor end.

For example:

The machine tool configuration is GSK25Ti and GR2000, using ISC system programming, where the X-axis is diameter programming, the lead is **6mm**, the Z-axis is radius programming, the lead is **8mm**, and the motor is directly connected to the X and Z-axis screws ($Z_M:Z_D= 1:1$), using a **25-bit** absolute encoder (the number of pulses per revolution of the encoder is **335,544,32**) to calculate the gear ratios corresponding to the X and Z axes.

- **Calculation of X-axis:**

Minimum ^{movement} unit: 0.00005mm (ISC system, diameter programming)

Servo Gear Ratio Setting

The setting value of PA29 is 16384.

The setting value of PA30 is 1875.

The setting value of PA33 is 32.

The setting value of PA34 is 1.

$$\text{Servo electronic gear ratio} = \text{Minimum movement unit} \times \frac{\text{Pulses per revolution of pulse encoder}}{\text{Lead}} \times \frac{Z_M}{Z_D} \times \frac{\text{CMR}}{\text{DMR}}$$

$$\frac{16384 \times 32}{1875} = 0.00005 \times \frac{33554432}{6} \times \frac{1}{1} \times \frac{1}{1}$$

● **Calculation of Z-axis:**

Minimum moving unit: 0.0001mm (ISC system, radius programming)

Servo Gear Ratio Setting

The setting value of PA29 is 16384.

The setting value of PA30 is 625.

The setting value of PA33 is 16.

The setting value of PA34 is 1.

$$\text{Servo electronic gear ratio} = \text{Minimum movement unit} \times \frac{\text{Pulses per revolution of pulse encoder}}{\text{Lead}} \times \frac{Z_M}{Z_D} \times \frac{\text{CMR}}{\text{DMR}}$$

$$\frac{16384 \times 16}{625} = 0.0001 \times \frac{33554432}{8} \times \frac{1}{1} \times \frac{1}{1}$$

2.4.2 Gear ratio setting

$$\text{Electronic Gear Ratio of CNC System} = \frac{\text{Command multiplication ratio (CMR: parameter No.1820)}}{\text{Detection multiplication ratio (DMR: parameter No.1816)}}$$

$$\text{Servo electronic gear ratio} = \frac{\text{Position pulse command multiplication coefficient (PA29} \times \text{PA33)}}{\text{Position pulse command differential coefficient (PA30} \times \text{PA34)}}$$

When the numerator of the electronic gear ratio (CMR/DMR) on the CNC side is greater than the denominator, the maximum speed allowed by the CNC will decrease. When the numerator of the electronic gear ratio is smaller than the denominator, the positioning accuracy of CNC may decrease. In order to ensure the positioning accuracy and speed index of CNC, when supporting digital servo with electronic gear ratio function, it is recommended to set the electronic gear ratio of CNC to 1:1, and set the calculated electronic gear ratio to the digital servo.

2.5 Servo related settings and adjustments

2.5.1 CNC servo parameter setting

After the system and the servo are correctly connected and powered on, correctly set the type of encoder, the direction of pulse output of each axis and the direction of axis movement, etc. The specific setting steps are as follows:

- Note 1: In automatic or MDI mode, when the direction of the commanded moving axis is opposite to the actual axis feed direction, modify the axis corresponding to parameter 1811#2.
- Note 2: In the manual mode, when the manual movement direction is opposite to the actual axis feed direction (the axis movement direction is correct in the automatic or input mode), modify the PLC data parameters K8.0~K8.4 corresponding to the axis.

Relevant Parameters			
	Parameter Number	Bit	Parameter Signification
Parameters	1811	#2	Pulse output direction selection for each axis 0: No negation 1: Negation
	1815	#5	Position detector 0: Absolute position detector is not used 1: Absolute position detector is used
PLC Data Parameters	K8.0~K8.4		Whether the manual movement direction of each feed axis is reversed 0: No negation 1: Negation

The larger the acceleration and deceleration time constant, the slower the acceleration and deceleration process, the smaller the impact of the machine tool movement, and the lower the processing efficiency; the smaller the acceleration and deceleration time constant, the faster the acceleration and deceleration process, and the greater the impact of the machine tool movement. The higher the efficiency at the time.

101

lower the processing efficiency.

The principle of acceleration/deceleration adjustment is to appropriately reduce the acceleration /deceleration time constant and increase the start/stop speed of acceleration and deceleration on the premise that the drive unit does not alarm, the motor does not lose step and the machine tool movement has no obvious impact, so as to improve the processing efficiency. . If the acceleration/deceleration time constant is set too small, and the acceleration/deceleration start/end speed is set too high, it is easy to cause the drive unit to alarm, the motor is out of step, or the machine tool vibration.

Setting of related parameters in the following situations:

- Fast runtime

Parameter	Parameter Signification
1420	Rapid traverse speed of each axis
1421	The lowest speed of each axis rapid traverse override (F0)
1601#4	During fast running, the program segment 0: no overlap (accurately in place) 1: overlap (smooth transition)
1610#2	Rapid feed acceleration/deceleration type 0: Linear acceleration and deceleration 1: S-type acceleration and deceleration
1620	Linear acceleration/deceleration time constant T for each axis' rapid traverse

- When manual MPG feed

Parameter	Parameter Signification
1423	Manual feed speed of each axis
1424	Manual rapid traverse speed of each axis
1434	The maximum feed rate of manual MPG feed for each axis
1610#4	The acceleration and deceleration of manual feed is 0: Exponential acceleration and deceleration 1: Linear acceleration and deceleration after interpolation
1625	FL speed of exponential acceleration/deceleration for manual feed of each axis
1630	Acceleration and deceleration time constant of MPG feed for each axis

- Reference point return

Parameter	Parameter Signification
1425	FL speed of each axis reference point return
1428	Reference point return speed of each axis

- Thread cutting

Parameter	Parameter Signification
1466	The maximum speed when performing thread cutting's run-out action
1467	Speed override when executing thread cutting run-out action
1469	Check the fluctuation of spindle speed when starting thread cutting
1610#5	The acceleration and deceleration of the run-out action during thread cutting are adopted 0: Exponential acceleration and deceleration 1: Linear acceleration and deceleration
1610#7	The speed override of the run-out action during thread cutting 0: invalid 1: valid
1626	Exponential acceleration/deceleration time constant during thread cutting cycle of each axis
1627	FL speed of exponential acceleration/deceleration in each axis thread cutting cycle
1628	The acceleration and deceleration time constant of the retracting action during the thread cutting cycle of each axis

Note: ◇When parameter 1601#4=0, at the intersection of the cutting feed trajectory, the feed rate should be reduced to the initial speed of acceleration and deceleration, and then accelerated to the command speed of the adjacent block, which is realized at the intersection of the trajectory accurate positioning, but will reduce the processing efficiency. When the parameter 1601#4=1, the adjacent cutting paths directly make a smooth transition by acceleration and deceleration. The feed speed may not necessarily drop to the starting speed at the end of the previous path, and an arc-shaped transition is formed at the intersection of the paths. (Inaccurate positioning), this kind of path transition method has good surface finish and high processing efficiency.

Table 2-9

Relevant Parameters:		
Parameter Number	Bit	Parameter Signification
1420		Rapid traverse speed of each axis
1421		The lowest speed of each axis rapid traverse override (F0)
1422		Maximum cutting feed rate of all axes
1423		Manual feed speed of each axis
1424		Manual rapid traverse speed of each axis
1425		FL speed of each axis reference point return
1428		Reference point return speed of each axis
1434		The maximum feed rate of manual MPG feed for each axis

Relevant Parameters:		
Parameter Number	Bit	Parameter Signification
1466		The maximum speed when performing thread cutting's run-out action
1467		Speed override when executing thread cutting run-out action
1469		Check the fluctuation of spindle speed when starting thread cutting
1601	#4	During fast running, the program segment 0: no overlap (accurately in place) 1: overlap (smooth transition)
1610	#2	Rapid feed acceleration/deceleration type 0: Linear acceleration and deceleration 1: S-type acceleration and deceleration
1610	#4	The acceleration and deceleration of manual feed is 0: Exponential acceleration and deceleration 1: Linear acceleration and deceleration after interpolation
1610	#5	The acceleration and deceleration of the run-out action during thread cutting are adopted 0: Exponential acceleration and deceleration 1: Linear acceleration and deceleration
1610	#7	The speed override of the run-out action during thread cutting 0: invalid 1: valid
1620		The linear acceleration/deceleration of each axis fast forward is always T
1622		The acceleration and deceleration time constant of the cutting feed after interpolation of each axis
1624		The acceleration and deceleration time constant of each axis manual feed after interpolation
1625		FL speed of exponential acceleration/deceleration for manual feed of each axis
1626		Exponential acceleration/deceleration time constant during thread cutting cycle of each axis
1627		FL speed of exponential acceleration/deceleration in each axis thread cutting cycle
1628		The acceleration and deceleration time constant of the retracting action during the thread cutting cycle of each axis
1630		Acceleration and deceleration time constant of MPG feed for each axis

2.7 Reference point and soft limit

The system supports three mechanical zero point establishment (also called reference point setting) methods, non-block type reference point setting, block type reference point setting and absolute encoder reference point setting.

Table 2-10

Reference Point Setting Mode	System parameter setting
Absolute encoder reference point setting	Parameter 1815#5 (APCx) is set to 1
Non-stop reference point setting	Parameter 1815#5 (APCx) is set to 0; Parameter 1002#1 (DLZ) is set to 1 or Parameter 1005#1 (DLZx) is 1 (set either of the two to 1)
Stopper-type reference point setting	Parameter 1815#5 (APCx) is set to 0; Parameter 1002#1 DLZ is set to 0 and parameter 1005#1 DLZx is set to 0

Note 1: When using an absolute encoder, after the reference point is established, the system will automatically save the reference point position after power off, and there is no need to set the reference point position again when the power is turned on next time.

Note 2: When using the non-block/block-type reference point setting, the reference point setting operation must be performed to establish the reference point every time the system is powered on.

Table 2-11

Relevant Parameters			
	Parameter Number	Bit	Parameter Signification
parameters	1005	#0	When the reference point is not established, in the automatic operation (MEM, DNC or input), when a movement command other than G28 is specified, will the system give an alarm? 0: Alarm 1: No alarm
	1006	#5	Set the direction of each axis returning to the reference point 0: Positive direction 1: Negative direction
	1201	#2	After the manual reference point return is completed, the local coordinate system 0: Do not cancel 1: Cancel
	1240		Set the coordinate value of the first reference point in the machine coordinate system
	1241		Set the coordinate value of the second reference point in the machine coordinate system
	1242		Set the coordinate value of the 3 rd reference point in the machine coordinate system
	1243		Set the coordinate value of the 4 th reference point in the machine coordinate system
	1425		Set the speed of each axis after deceleration when returning to the reference point (FL speed)
PLC data parameters	K12.2		Whether the direction keys of the zero return operation are self-locking 0: No self-locking 1: Self-locking

2.7.1 Reference point setting of absolute encoder

When the machine tool is equipped with an absolute encoder, when the system has not established a reference point, it is necessary to perform the operation of returning to the reference point of the absolute encoder. When the tool returns to the reference point, the end of the return to reference point LED light is on, and the coordinate system is automatically set.

Operation method of absolute encoder returning to reference point:

- ① In the "manual" or "MPG" mode, move the machine tool to the reference point position to be set.
- ② Set the parameter 1815#4 (APZx) to 0, the system will generate an alarm, turn off the power and then turn on the power again.
- ③ Press the return to reference point switch (turn the method selection knob to the "zero return" position).



- ④ Press the corresponding feed axis to return to the reference point to execute the operation of returning to the reference point.

- ⑤ The reference point return completion light (LED) of the corresponding axis is on and the reference point return is over. At this time, the system saves the current encoder position and automatically sets parameter 1815#4 (APZx) to 1.

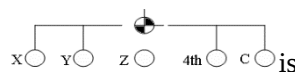


Table 2-12

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
1815	#4	When using an absolute encoder, the position of the mechanical position and the absolute position detector 0: inconsistent 1: consistent
1815	#5	Position detector 0: Do not use the absolute position detector 1: Use the absolute position detector

2.7.2 Setting of reference point with stop

When the system sets the function of returning to the reference point without a block as invalid (that is, the setting of the reference point with a block is valid), the machine tool needs to install a deceleration switch to realize the function of returning to the reference point.

Operation method of reference point return with a stop:

- ① Confirm that the overtravel limit switch is valid.

- ② Press the back to reference point button to switch the working mode of the system to back to reference point (turn the mode selection knob to the "return to zero" gear);



- ③ In order to reduce the zero return speed, first adjust the rapid-travel override switch to the low-speed gear position;
- ④ Press the direction selection switch of the corresponding feed axis to return to the reference point, the system reads in the corresponding axis and direction selection signal, and starts the operation of returning to the reference point.
- ⑤ The carriage moves to the deceleration point at a fast moving speed, the deceleration signal DECx is valid (the effective level of the signal is set by parameter 3009#5 (DECx)), after deceleration, it continues to move to the reference point at the zero return low speed set in parameter 1425 .
- ⑥ After the tool leaves the deceleration switch position and the deceleration signal DECx changes, the system starts to detect the motor one-revolution signal nPC.
- ⑦ When the system detects the first nPC signal of the motor, it sets the reference point return end signal ZPx and reference point establishment signal ZRFx to 1, the reference point return completion light

(LED) x y z 4th c lights up, and the reference point return ends.

Note: The mechanical zero stop is usually installed at the maximum stroke, and the effective stroke of the zero return stop is above 25 mm. Enough deceleration distance must be ensured to ensure that the speed can be lowered to ensure accurate zero return. The faster the mechanical zero return is performed, the longer the zero return block, otherwise the moving carriage will rush through the zero return block due to CNC acceleration and deceleration, machine inertia, etc., and there is not enough deceleration distance, which will affect the accuracy of zero return. In addition, before returning to zero, make sure that the carriage will not interfere with other parts of the machine tool during the zero return movement to ensure safety.

Usually the connection method of the supporting AC servo motor: use a limit switch and a schematic diagram of the one-turn signal of the servo motor respectively.

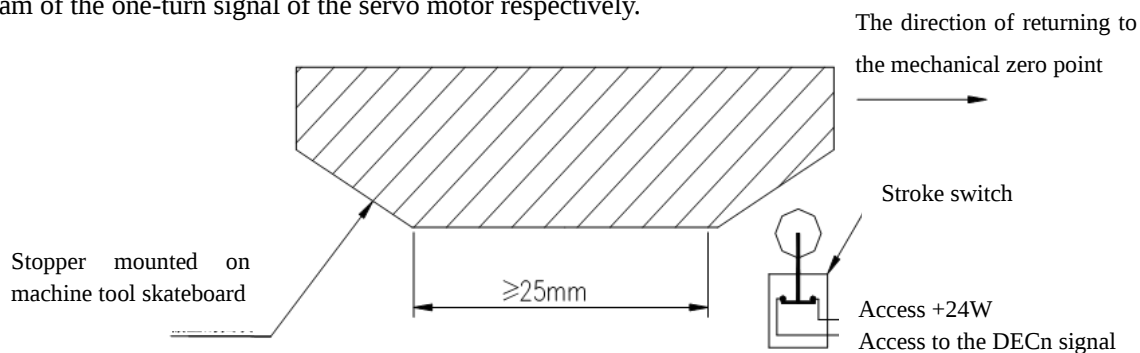


Fig. 2-3

With this connection method, when the deceleration switch is released when returning to the mechanical zero point, avoid the one-turn signal of the encoder at the critical point after the release of the limit switch, and ensure that the one-turn signal of the encoder reaches the encoder's one-turn signal after half a revolution of the motor. Zero precision. The stop position can be fine-tuned to reduce the zero return error.

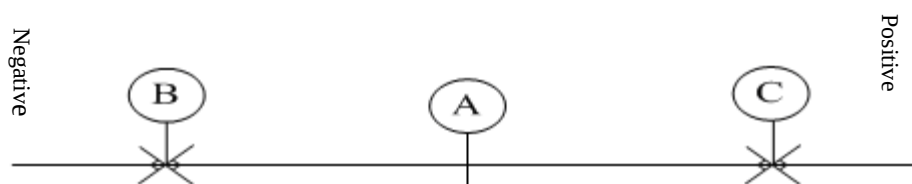
2.7.3 Setting of reference point without stopper

When the system sets the function of returning to the reference point without a stopper as valid, the machine tool can realize the reference point return without installing a deceleration switch. When the tool returns to the reference point, the end of the return to reference point LED light is on, and the coordinate system is automatically set.

Operation method of reference point return without stopper:

- ① The machine tool feeds along the axis in the return direction of the reference point, so that it stops near the reference point, but does not exceed the reference point position.
- ② Press the return to reference point switch (turn the method selection knob to the "zero return" position).
- ③ Press the direction selection switch of the corresponding feed axis for returning to the reference point, the corresponding axis and direction selection signal Jx is set to 1, and the operation of returning to the reference point is started.
- ④ The tool moves to the reference point in the direction set by parameter 1006#5 (ZMIx) and the FL speed set in parameter 1425.
- ⑤ When the system detects the first PC signal of the motor, set the reference point return end signal ZPx and reference point establishment signal ZRFx to 1, the reference point return completion light (LED)

x ○ y ○ z ○ 4th ○ C ○ lights up, and the reference point return ends.



- A: Perform return to the position before the reference point without a block;
- B: The position of the reference point after returning to the reference point in the negative direction, that is, the place where the first PC signal is generated after the point A moves in the negative direction;
- C: The position of the reference point after returning to the reference point in the positive direction, that is, the place where the first PC signal is generated after the point A moves in the positive direction.

Table 2-13

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
1002	#1	Whether the non-stop reference point setting function is valid 0: Invalid 1: Valid (valid for all axes)
1002	#3	G28 command when the reference point is not established 0: Same as manual return to reference point, use deceleration block to return to reference point 1: P/S alarm occurs
1005	#1	Whether the non-stop reference point setting function is valid 0: invalid 1: valid
1300	#6	After LZR is powered on and before manually returning to the reference point, whether to perform the first stored stroke detection 0: proceed 1: not proceed

2.7.4 Stored stroke check settings

The system provides three stored stroke inspection areas: stored stroke limit check 1, stored stroke limit check 2, and stored stroke limit check 3. These three specified areas cannot be entered by the tool.

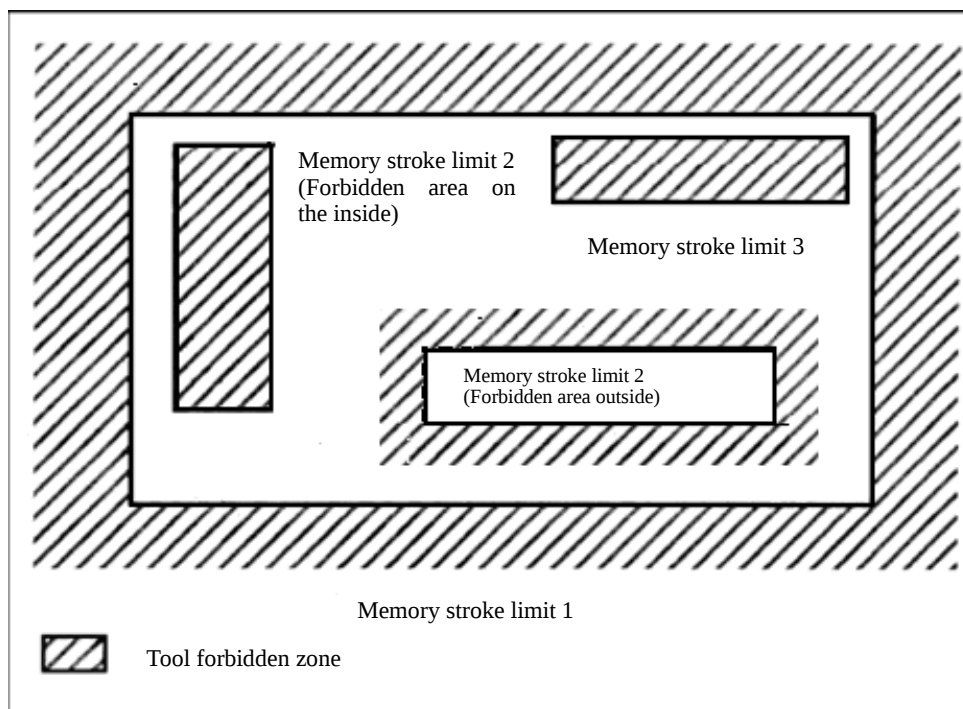


Fig. 2-4

Stored stroke limit check 1:

The boundary is set by parameters 1320, 1321 or parameters 1326, 1327, and the outside of the set boundary is the forbidden zone. The machine tool manufacturer usually sets this area as the maximum stroke range.

Note 1: Parameter 1300#7 (BFA) is only valid for stroke limit check 1.

Note 2: When parameter #1300.7 is set to 1, during "program execution", before executing the current program block, the system will pre-judge whether the trajectory after executing the program block will exceed the stored stroke. If it exceeds, an overtravel alarm will be triggered; otherwise, the program block will continue to be executed.

Note 1: When parameter #8400.6 is set to 1, the additional axis' stored stroke detection 1 is activated. During "program execution", before executing the current program block, the system will pre-judge whether the trajectory after executing the program block will exceed the stored stroke. If it exceeds, an overtravel alarm will be triggered; otherwise, the program block will continue to be executed.

Stored stroke limit check 2 (G22 G23):

The boundary is set by parameters 1322, 1323 or commands, and parameter 1300#0 sets whether the prohibited area is inside or outside the set boundary. When programming, use G22 instruction to prohibit the tool from entering the forbidden area, and G23 instruction to allow the tool to enter the forbidden area. G22 and G23 should be designated separately in the program and are independent blocks. Refer to the G instruction section for details.

Stored stroke limit check 3:

The boundary is set by the parameters 1324 and 1325, and the inside of the 3 boundary is set as the forbidden zone.

Note: When setting the restricted area, if the two points are set to be the same, the area is as follows:

1. When the stored stroke check 1 is in the forbidden area, if the two points are set to be the same, all the areas are the forbidden area.
2. When the stored stroke check 2 or stored stroke check 3 is in the setting forbidden area, if the two points are set to be the same, all areas are movable areas.

Display alarm time:

Parameter 1300#7 (BFA) selects whether to display the alarm immediately before the tool enters the forbidden area or display the alarm immediately after the tool enters the forbidden area.

Overtravel alarm release:

When the tool exceeds the stored stroke limit, an alarm is displayed and the tool decelerates and stops. Switch to manual mode to move the tool in the reverse direction (such as positive overtravel, move out in

negative direction; if negative overtravel, move out in positive direction) Forbidden zone, press the reset button to clear the alarm display.

Table 2-14

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
1300	#0	The forbidden area of stored stroke detection 2 set with parameters (No.1322, No.1323) is 0: Inner area 1: Outer area
1300	#2	Whether the stored stroke detection switching signal EXLM is valid 0: invalid 1: valid
1300	#5	Stroke detection 3 release signal RLSOT3 is valid 0: invalid 1: valid
1300	#6	Whether to perform the first stored stroke detection after turning on the power supply and before manually returning to the reference point 0: proceeding 1: not proceeding
1300	#7	When a command that exceeds the stored stroke is issued 0: Alarm occurs after the trip is exceeded 1: Alarm occurs before the trip is exceeded
1310	#0	Whether each axis is checked for stored stroke detection 2 0: not proceeding 1: proceeding
1310	#1	Whether each axis is checked for stored stroke detection 3 0: not proceeding 1: proceeding
1320		Coordinate value of the positive direction boundary of each axis stored stroke detection (PC1)
1321		Coordinate value of the positive direction boundary of each axis stored stroke detection 1 (NC1)
1322		Coordinate value of the positive direction boundary of each axis stored stroke detection 2 (PC2)
1323		Coordinate value of the positive direction boundary of each axis stored stroke detection 2 (NC2)
1324		Coordinate value of the positive direction boundary of each axis stored stroke detection 3 (PC3)
1325		Coordinate value of the positive direction boundary of each axis stored stroke detection 3 (NC3)
1326		Coordinate value II of the positive direction boundary of each axis stored stroke detection (PC12)
1327		Coordinate value II of the negative direction boundary of each axis stored stroke detection (NC12)

2.8 Error compensation function

2.8.1 Stored pitch error compensation

By setting the pitch error compensation data, each axis can perform pitch error compensation in its detection unit.

Stored pitch error compensation setting steps:

● Parameter setting:

Set the minimum position number and maximum position number of the pitch error compensation of each axis through the parameters No.3621 and No.3622 to determine the range of the compensation table for each axis. The compensation position in the pitch error compensation table corresponding to each machine zero point is called the pitch error compensation origin (reference point); the pitch error compensation origin is set by the data parameter No.3620. According to actual needs, the compensation origin can be set at any position within the compensation table of each axis.

Parameter	Significance
3600#0	Whether to set pitch error compensation 0: Use 1: Do not use
3600#3	Pitch error compensation display 0: Horizontal 1: Vertical
3620	The position number of the pitch error compensation for each axis at the reference point
3621	Minimum position number for pitch error compensation of each axis
3622	Maximum position number for pitch error compensation of each axis
3623	Magnification of pitch error compensation for each axis
3624	Position interval for pitch error compensation of each axis

● Define compensation position:

In order to specify the compensation position for each axis, the positive and negative movement directions of compensation should be specified based on the reference point. If the stroke of the machine tool exceeds the specified range in the positive or negative direction, the pitch error compensation will not work if the range is exceeded.

On the setting screen of pitch error compensation, a total of 1024 compensation positions from 0 to 1023 can be used. The available parameters can arbitrarily assign position numbers to each axis. The compensation position number of the reference point (parameter 3620), the minimum compensation position number (parameter 3621) and the maximum compensation position number (parameter 3622) must be set for each axis.

● Set Pitch error compensation value:



Under the system page set, press the  soft key to enter the pitch compensation page, as shown in Fig. 2-5.

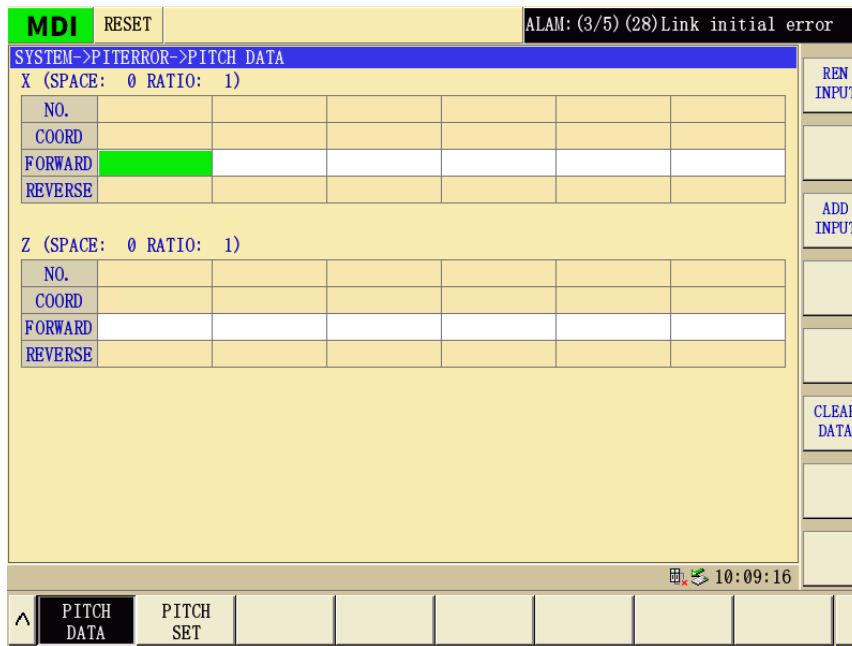


Fig. 2-5

- ① On this page, the user can view and set the pitch compensation value corresponding to each pitch compensation number.
- ② Under the pitch compensation page, select the compensation value of the axis to be set through the

page key  , and cursor movement keys , ,  and .

- ③ When the operation authority level is equal to or higher than [2] level, press  the key to make the compensation value of the selected axis in a modifiable state. Enter the compensation value through the numeric keys, and then press  the key to complete the modification.

- ④ Clicking the  or  button allows you to import the pitch compensation configuration files from a USB flash drive.

Note: The compensation value of the pitch error and the distance between the pitch error compensation points are related to whether the axis is diameter programming or radius programming (Parameter No.1006#3). If the axis is diameter programming, the parameter setting value is the diameter value; if it is radius programming, the setting value is the radius value, and the unit is the detection unit.

Example 1: Linear axis pitch error compensation

Assuming it is a metric machine tool, ISB incremental system, the machine stroke is -400mm~+400mm, the pitch error compensation position interval is 80mm, and the compensation position number of the reference point is 23, then:

The number of the farthest compensation position in the negative direction

$$= \text{the compensation position number of the reference point} - \left(\frac{\text{machine stroke in the negative direction}}{\text{compensation position interval}} \right)$$

$$= 23 - 400/80 + 1 = 19$$

The farthest compensation position number in the positive direction

$$= \text{the compensation position number of the reference point} + \left(\frac{\text{machine stroke in the positive direction}}{\text{compensation position interval}} \right)$$

$$= 23 + 400/80 = 28$$

Therefore, the setting values of parameters 3620~3624 are as follows.

Parameter	Setting Value	Note
3620	23	The position number of the pitch error compensation for each axis at the reference point
Parameter	Setting Value	Note
3621	19	Minimum position number for pitch error compensation of each axis
3622	28	Maximum position number for pitch error compensation of each axis
3623	1	Magnification of pitch error compensation for each axis
3624	80	Position interval for pitch error compensation of each axis

The corresponding relationship between the machine coordinate and the compensation position number is shown in Fig. 2-6.

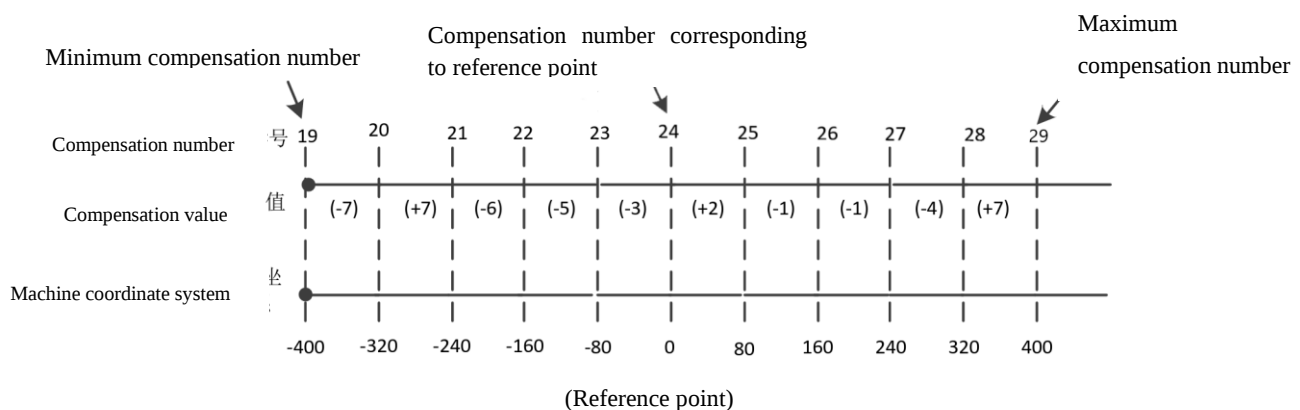


Fig. 2-6

According to Fig. 2-6.

Machine Coordinate System	Compensation Parameter Number	Compensation Amount
-400 ~ -320	19	-7
-320 ~ -240	20	+7
-240 ~ -160	21	-6
-160 ~ -80	22	-5
-80 ~ 0	23	-3
Reference point 0		
0 ~ 80	24	+2
.....		...

Example 2: Definition of pitch error compensation position of rotary axis

Assuming it is a metric machine tool, ISB incremental system, the amount of movement per revolution is 360°, the pitch error compensation position interval is 45°, and the compensation position number of the reference point is 80, then

The number of the farthest compensation position in the negative direction

= the compensation position number of the reference point + 1

= 80+1=81

The farthest compensation position number in the positive direction

= the compensation position number of the reference point - $\left(\frac{\text{movement per revolution}}{\text{compensation position interval}} \right)$

= 80+360/45=88

Therefore, the setting values of parameters 3620~3624 are as follows.

Parameter	Setting Value	Note
3620	80	Pitch error compensation position number of each axis at reference point
3621	81	Pitch error compensation number at the farthest negative end of each axis
3622	88	Pitch error compensation number at the farthest positive end of each axis
3623	1	Pitch error compensation magnification of each axis
3624	45	Pitch error compensation position interval of each axis

The corresponding relationship between the machine coordinate and the compensation position number is shown in Fig. 2-7.

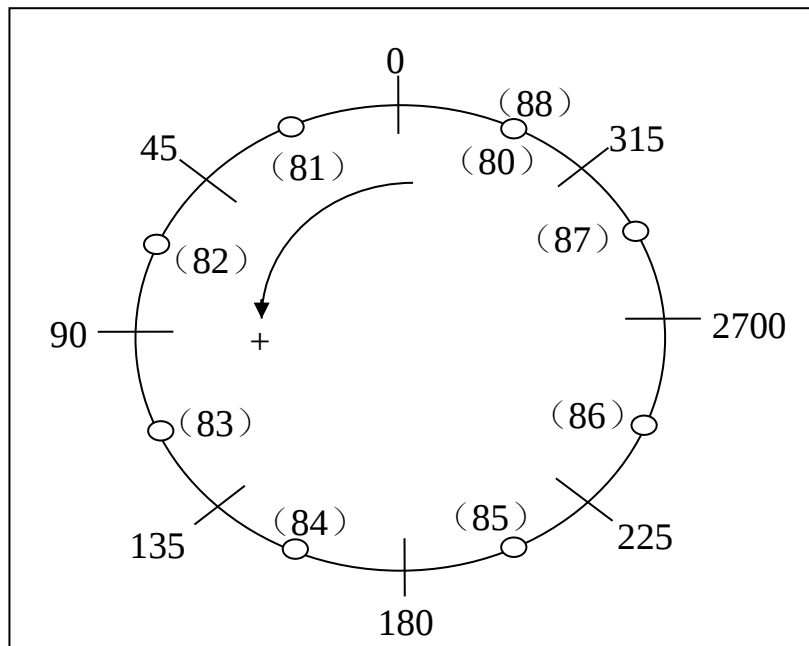


Fig. 2-7

If the sum of the compensation values from positions 81 to 88 is not 0, position deviation will occur. The so-called sum refers to the accumulation of the pitch error compensation value per revolution. In addition, the compensation position of 80 and 88 must be set to the same compensation value.

For example:

Compensation location number	80	81	82	83	84	85	86	87	88
Set compensation value	+1	-2	+1	+3	-1	-1	-3	+2	+1

Table 2-15

Parameter Number	Parameter Signification
3620	Pitch error compensation number of each axis reference point
3621	The number of the pitch error compensation point at the farthest end in the negative direction of each axis
3622	The number of the pitch error compensation point at the farthest end in the positive direction of each axis
3623	Pitch error compensation magnification of each axis
3624	The pitch error compensation point spacing of each axis
3628	Setting value of pitch compensation pulse frequency

2.8.2 Double-directional pitch error compensation

The stored pitch error compensation does not distinguish the direction of movement. However, in the two-way pitch error compensation, you can distinguish and set the amount of pitch error compensation when moving in the positive direction and when moving in the negative direction, and perform pitch error compensation in different directions. In addition, when the movement is reversed, the compensation amount is automatically calculated based on the pitch error compensation data, and the compensation when the movement direction is reversed is the same as the backlash compensation. As a result, the path difference between the positive direction and the negative direction can be further reduced.

To use the two-way pitch error compensation function, set the two-way pitch error compensation function to be valid (parameter No.3605#0="1").

Two-way pitch error compensation setting steps:

- **Parameter setting**

Parameter Number	Content
3605#0	Whether to use two-way pitch error compensation 0: not used 1: used
3620	Pitch error compensation number of each axis reference point
3621	The number of the pitch error compensation point at the farthest end in the negative direction of each axis
3622	The number of the pitch error compensation point at the farthest end in the positive direction of each axis
3623	Set the magnification of each axis's pitch error compensation
3624	The pitch error compensation point spacing of each axis
3627	The compensation value of the pitch error in the reference point when moving from the direction opposite to the return-to-origin direction to the reference point

- **Set compensation value**

It is the same as the stored pitch error compensation function, but when setting the pitch error compensation data, always set the data value seen from the negative side of the coordinate system. The pitch error data in the negative direction must be the same as the number of set points of the pitch error compensation data in the positive direction. For the pitch error data in the negative direction, the data value seen from the negative side of the coordinate system is always set.

An example is shown in Fig. 2-8.

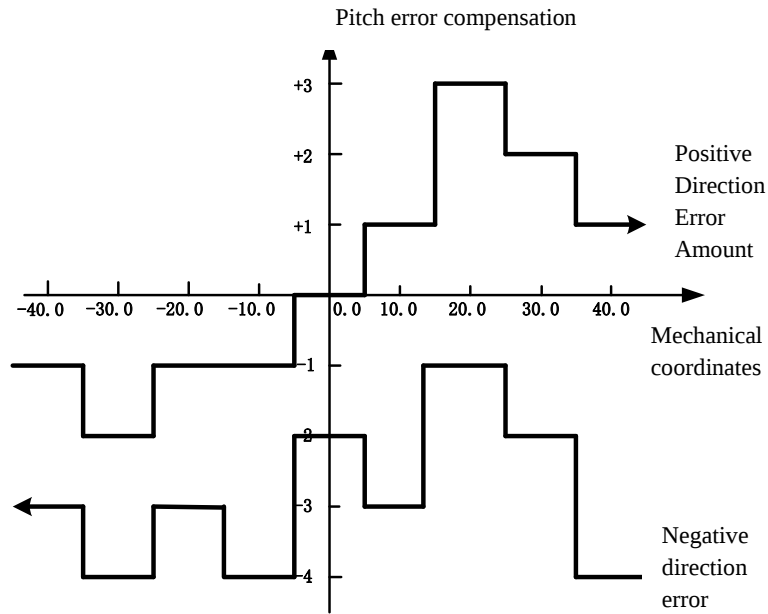


Fig. 2-8

From Fig. 2-8, we can get:

Pitch error compensation data:

Compensation point number	20	21	22	23	24	25	26	27
Positive direction	-1	+1	0	+1	+1	+2	-1	-1
Negative direction	-1	+1	-1	+2	-1	+2	-1	-2

Setting value of each parameter:

Table 2-16

Parameter Number	Setting Value	Content
3605#0	1	Whether to use two-way pitch error compensation 0: not used 1: used
3620	23	Pitch error compensation number of each axis reference point
3621	20	The number of the pitch error compensation point at the farthest end in the negative direction of each axis
3622	27	The number of the pitch error compensation point at the farthest end in the positive direction of each axis
3623	1	Set the magnification of each axis's pitch error compensation
3624	100	The pitch error compensation point spacing of each axis
3627	-2	The compensation value of the pitch error in the reference point when moving from the direction opposite to the return-to-origin direction to the reference point

2.8.3 Backlash compensation

Due to the error of the transmission mechanism, when the machine tool moves in the reverse direction, a part of the movement will be lost, which will affect the machining accuracy. In order to reduce the influence of the error caused by the reverse movement during processing, the system provides a backlash error compensation function.

The compensation value of the axis backlash is related to whether the axis is diameter programming or radius programming (straight/radius programming directly affects the minimum movement unit of the system). If the axis is diameter programming, the parameter setting value is the diameter value; if it is radius programming, the setting value is the radius value, and the unit is the detection unit.

$$\text{Detection unit} = \frac{\text{Minimum moving unit}}{\text{Command multiplier ratio (CMR)}}$$

Backlash compensation must be accurately compensated to improve the accuracy of the processing. You can use a dial indicator, a dial indicator or a laser detector for measurement. In order to make accurate compensation, it is not recommended to use MPG or single-step method to measure the screw backlash. It is recommended to measure the backlash as follows:

Editing program:

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

- ① The backlash error compensation value should be set to zero before measurement;
- ② Single-segment running program, find the measurement datum 1 point after positioning twice, record the current data, and run 1mm in the same direction, and then run 1mm to 2 points in the reverse direction to read the current data.

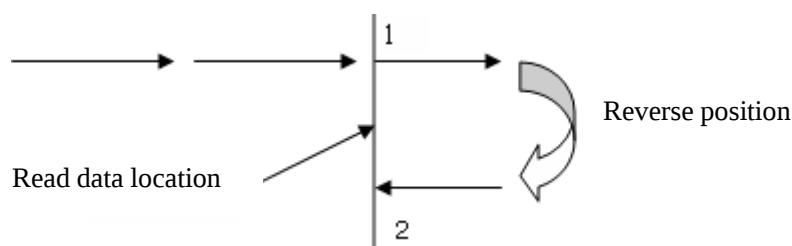


Fig. 2-9 Schematic diagram of backlash measurement method

- ③ Backlash error compensation value = | 1 point recorded data – 2 points recorded data|; Convert the calculated data into the detection unit and then input it into CNC data parameter No.1851.

Data 1: The data of the dial indicator is read at 1 place;

Data 2: The data of the dial indicator can be read in 2 places;

Detection unit=Minimum moving unit/CMR;

For example: when the system is set to IS-C system by parameter (parameter No.1004#1 ISC is set to 1), metric machine tool (parameter No.1001#0 INM is set to 0), if parameter No.1820 (used to set each The axis command multiplication ratio) is set to 1, so the system command multiplication ratio CMR=1;

Therefore:

X-axis: detection unit = minimum movement unit / CMR = 0.00005mm / 1 = 0.00005 mm;

Z-axis: detection unit = minimum movement unit / CMR = 0.0001mm / 1 = 0.0001 mm;

If the backlash error compensation value of the X-axis measured by the dial indicator is 0.0150mm, set the parameter No.1851 to 300; the backlash error compensation value of the Z-axis measured by the dial indicator is 0.0300mm , Set the parameter No.1851 to 300.

Parameter setting steps of backlash are as follows:

- ① Measure the amount of backlash compensation according to the above method and save it in parameter 1851. Note that the parameter unit is the detection unit.
- ② After setting the backlash compensation value, set the output mode of backlash compensation according to parameter 1800#7 (BDEC): 0: fixed pulse frequency output 1: output according to acceleration and deceleration characteristics.
- ③ When parameter 1800#7 (BDEC) is set to 0 (fixed pulse frequency output), set the pulse output frequency during fixed pulse frequency output through parameter 1800#6 (BD8), 0: compensate with the set frequency 1: 1/8 compensation of the set frequency. The compensation setting frequency is set in parameter 1853.
- ④ When parameter 1800#7 (BDEC) is set to 1 (output according to acceleration and deceleration characteristics), the effective time constant of acceleration and deceleration can be set through parameter 1859.

Table 2-17

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
1800	#6	Pulse output frequency of backlash compensation 0: Compensate with the frequency set by parameter #1853 1: Compensate with 1/8 of the frequency set by parameter #1853
1800	#7	Backlash compensation method 0: Output at a fixed pulse frequency (set by parameters #1853 and #1800.6) 1: Pulse frequency is output according to acceleration and deceleration feature

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
1851		Backlash compensation amount of each axis
1852		Backlash compensation amount during rapid movement of each axis
1853		Setting value of backlash compensation pulse frequency
1859		Effective time constant of backlash acceleration and deceleration for each axis

2.9 Tapping function

2.9.1 Tapping method selection

The tapping cycle is divided into ordinary tapping and rigid tapping. Rigid tapping is further divided into standard rigid tapping cycle, high-speed deep hole rigid tapping cycle, and deep hole rigid tapping cycle. The specified methods are shown in the table below.

Tapping Method	Method of Specifying Tapping Mode
Ordinary tapping	Parameter No.5200#0 is set to 0 and M29 command is not specified
Rigid tapping	Specify M29 S_ in the previous section of the G84 (G88) tapping block (parameter No.5200#0 needs to be set to 0)
	Command M29 S_ in the same segment as the G84 (G88) tapping block (parameter No.5200#0 needs to be set to 0 and the C-axis clamping M code cannot be specified in the same segment)
	G84 (G88) is directly used as a rigid tapping command (parameter No.5200#0 needs to be set to 1)

Rigid Tapping Machining Mode	Ways to Specify Rigid Tapping Machining Mode
Standard rigid tapping cycle	Do not specify Q value or Q value 0
High speed deep hole rigid tapping cycle	Specify the Q value (not zero) and the parameter PCP (No.5200#5) = "0"
Deep hole rigid tapping cycle	Specify the Q value (not zero) and the parameter PCP (No.5200#5) = "1"

Note 1: Parameter 5210 can be used to specify the M code for rigid tapping. When it is specified as 0, the default is M29.

Note 2: If the system standard PLC program is used, the PLC parameter K10.4 must be set to 1 when using rigid tapping.

Note 3: For detailed tapping programming instructions, please refer to the corresponding CNC system programming and operation manual.

Note 4: If the system standard PLC program is adopted and the machine tool has a power head, CNC parameter No.5200#7 needs to be set to 1, and M24\M25\M26 is used to select which spindle to tap.

*M24	Select the first spindle rigid tapping	Function interlock, state maintenance
M25	Select the second spindle rigid tapping	
M26	Select the third spindle rigid tapping	

Table 2-18

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
5200	#0	Method of commanding rigid tapping 0: Command the M code of rigid tapping before commanding G84/G88 (see parameter No.5210) 1: Rigid tapping is not used for M code command, G84/G88 is used as G code for rigid tapping, ordinary tapping is not used
5200	#2	When specifying the command to cancel rigid tapping, whether the cancellation of rigid tapping waits for the rigid tapping signal RGTP to become 0: 0: Do not wait 1: Wait
5200	#4	Rigid tapping retract feedrate multiplier 0: Invalid 1: Valid (the override value is set in parameter No.5211)
5200	#5	When address Q is commanded during tapping cycle/rigid tapping 0: Used as a high-speed deep hole tapping cycle 1: Used as a deep hole tapping cycle
5200	#6	During rigid tapping: feed hold and single block operation 0: not allowed 1: allowed
5200	#7	In multi-spindle control, select the spindle for rigid tapping 0: Use spindle selection signal SWS1~SWS3 1: Use rigid tapping spindle selection signal RGTSP1~RGTSP3
5201	#2	Time constants for cutting and retracting in rigid tapping: 0: Same 1: Different
5201	#3	Set the unit of the parameter (No.5211) of the drawing override for rigid tapping 0: set to 1% 1: set to 10%
5201	#4	The spindle speed during drawing is commanded by the program (address J), so that the override is 0: invalid 1: valid
5202	#6	The command range of the drawing override command (address J) based on the rigid tapping program command is 0: 100%~200% 1: 100%~2000%

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
5203	#4	In rigid tapping, make the override and override cancellation signal based on the feedrate override selection signal 0: invalid 1: valid
5209	#0	In rigid tapping, the drilling axis 0: Select by plane selection 1: G84 is fixed as Z-axis and G88 is fixed as X-axis
5210		Set the M code that specifies the rigid tapping method. When it is set to 0, CNC considers it as M29.
5211		Set the multiplier value (0~200) for rigid tapping retraction, which is only valid when parameter DOV (NO.5200#4) is set to 1.
5213		Set the retraction amount of the high-speed deep hole tapping cycle or the empty stroke amount of the deep hole tapping cycle.
5241		Set the maximum spindle speed for rigid tapping.
5261		Time for the spindle to reach a speed of 1000r/min when it achieves rigidity (0~4000ms)
5271		Set the linear acceleration/deceleration time constant (0~4000ms) for the spindle and tapping axis during rigid tapping retraction.
5275		Set in G84/G88 ordinary tapping (non-rigid tapping), the movement of the tapping axis lags the number of compensation cycles sampled by the spindle encoder. Generally, it is appropriate to set it to 4~8.

2.9.2 Rigid tapping debugging settings

Before enabling the rigid tapping function, the following settings are required.

Note: The following PLC parameter settings are subject to the system standard ladder diagram.

System Cs settings

The method of controlling the rotation speed of the spindle is called spindle speed control (the spindle rotates through a speed command), and the control of the spindle position is called spindle contour control (the spindle rotates through a position command). The speed and position control function of the spindle is the CS contour control function. It can perform indexing and rigid tapping, and can interpolate and process contour curves with other feed axes.

- Cs Setting operation of the system is as follows:

1. Press  →  →  to enter the setting page.

MDI RESET hold key short circuit, or key wir

SETTING->SYSTEM SETTING->BASIC SETTING

AXIS SET

Axis num 3 Unit ISB Link Used

Axis	Extname	Type	Attribute	Radius	ID
Axis1	X	0	Linear	1	1
Axis2	Z	0	Linear	3	2
Axis3	C	0	Rotation A	0	3

SPINDLE SET Set the C-axis as type A rotary axis, and set the axis attribute to 0

Spindle num 1

Type	Axis	Max speed	ID
S1	Link cs	C	1000 3

IO SET Set the spindle type as a bus Cs axis, with the corresponding feed axis being the C-axis.

IO num 1

ID

14:40:13

^ BASIC SET GEAR RATIO REFERENCE STROKE LIMIT DATA OPTION

Fig. 2-10

2. Press  →  →  →  to enter the system setting page.

MDI RESET 0: (2/3) Machine panel devID error.

SYSTEM->LINK->COMMUNICATION->CNC SET

DEVICE NAME	CNC STN. ADDR	DEVICE STN. ADDR	DEVICE MODEL	LINK NO
X	0	--	--	--
Z	2	2	GR2030L	3
C	1	1	GR2030L	2
S	1	1	GR2030L	2
IO-1	100	100	IOR44T	1

Set the C-axis slave device to be consistent with the spindle slave device.

System parameters 1023: Set the station address for each axes (-31, -21, -11, -1, 0~99), Set to 0 to be invalid

14:45:29

^ CNC SET DEV SET

Fig. 2-11

Table 2-19

Basic parameters of spindle Cs contour control		
	Parameter number	Parameter definition
System Parameters	1010	If there is no C axis, the system will add an axis first, and the parameter needs to be re-powered after modification.
	8130	
	1020	Axis name: C axis is set to 67
	1022	Axis attribute, can be set to 0, if the C axis needs special interpolation, set it to XYZ parallel axis as needed
	1023	Set the servo axis number, which is the same as the corresponding spindle axis number #3717
	1004#6	Set the latest command increment of the rotary axis to 10 times, (that is, if the system is ISC equivalent, the equivalent of the C axis is 0.001 after this parameter is set to 1), after the parameter is set to 1, the gear ratio of the rotary axis is calculated The minimum equivalent is $\times 10$, which can increase the speed of the C axis.
	1006#0 1006#1	Set the C axis as the rotation axis: 00: Linear axis 01: Rotary axis (Type A) 11: Rotary axis (B type)
	1008#0	Set whether the cycle display function of the rotary axis is valid 0: Invalid 1: Valid
	1008#1	Set the axis rotation direction when absolute command 0: Rotation direction closer to the target 1: The direction specified by the command value symbol
	1260	Set the movement amount per revolution of the rotary axis. (ISB:1.0000~999999.999) (ISC:1.0000~99999.9999)
	1811#2	Pulse output direction selection for each axis 0: No negation 1: Negate
PLC Parameters	3701#7	CS axis contour control is valid, set to 1
	K10.4	Whether the rigid tapping function is valid, it must be set to 1
	K16.5	Whether CS axis switching control is valid, it must be set to 1

Power head debugging

- When the power head is controlled by analog voltage

1、 Press  →  →  to enter the setting page.

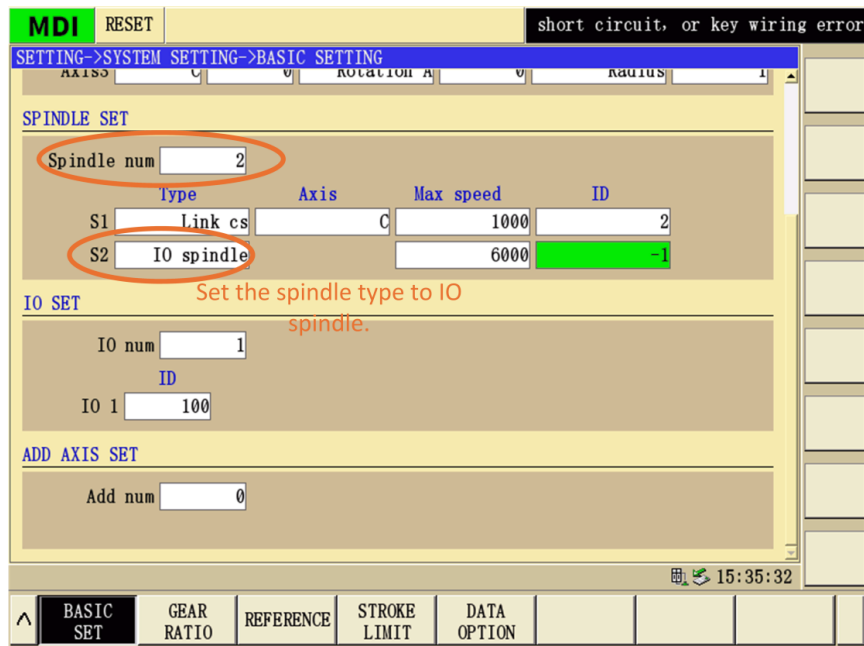


Fig. 2-12

2. Press  →  in turn to enter the parameter page.

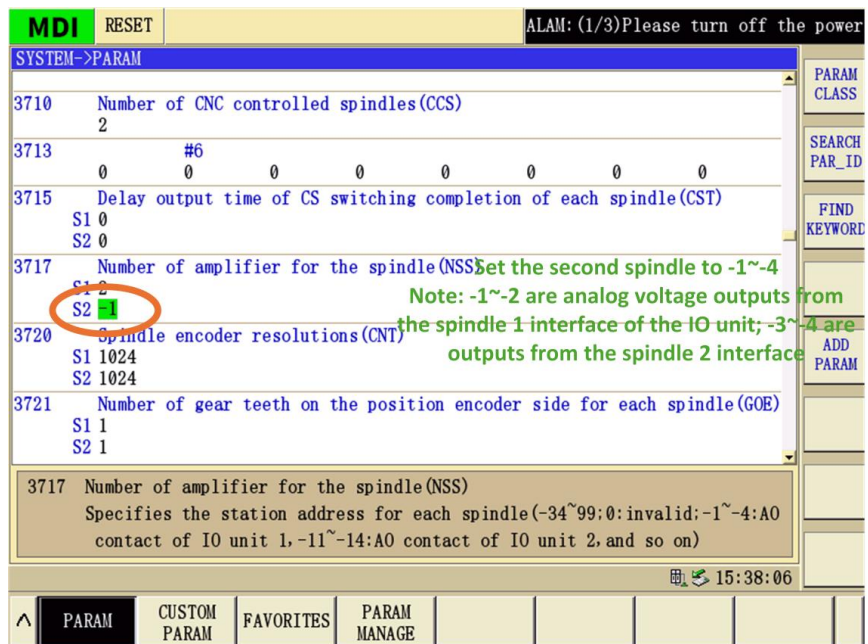


Fig. 2-13

● When the power head is bus control

1. Press  →  →  to enter the setting page.

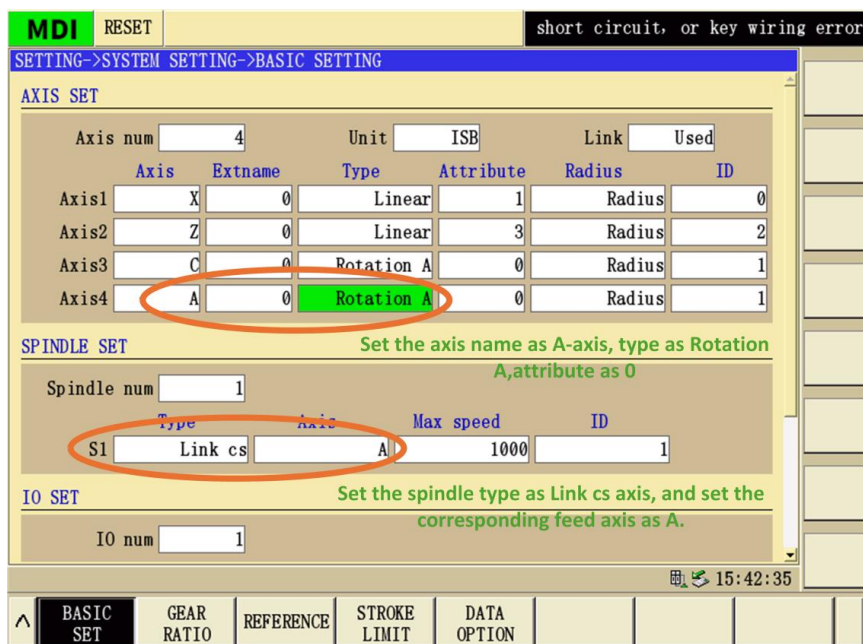


Fig. 2-14

2. Press  →  →  →  to enter the system setting page.

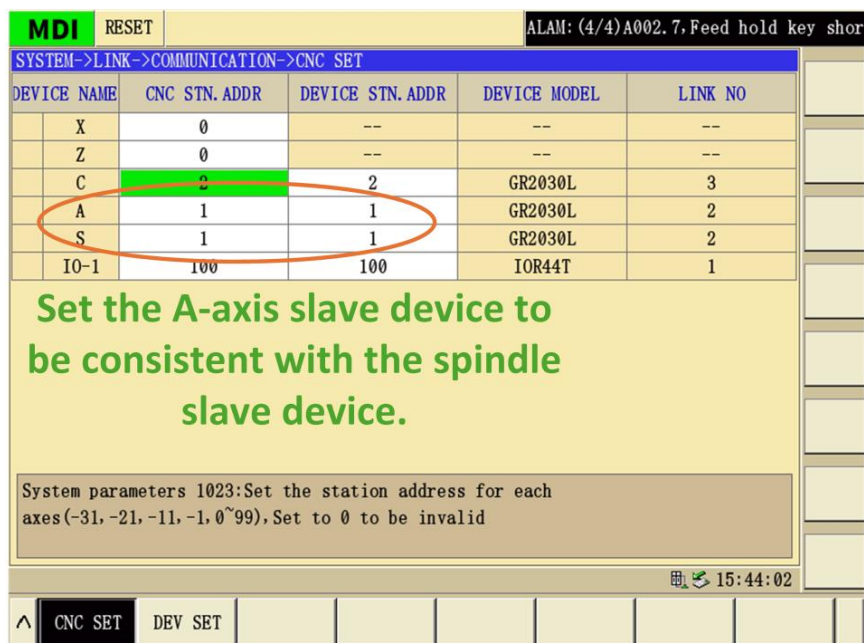


Fig. 2-15

Table 2-20

Related Parameters of the Second/Third Spindle			
	Parameter Number	Parameter Definition	Note
System parameters	3710	CNC control spindle number	Modify it to 2, if there is a second spindle, set it to 3. After modifying this parameter, the system must be powered on again
	3717	Each spindle amplifier number	Set 1~99 as main line spindle, -1~ -4 as analog spindle
	3723	Spindle encoder number	Set 0 as bus feedback, 1~2 as encoder interface CN21/CN22 interface feedback
	3705#4	Output SF code	Multi-spindle must be set to 1
	5200#7	In multi-spindle control, select the spindle for rigid tapping 0: Use spindle selection signal SWS1-SWS4 1: Use rigid tapping spindle selection signal RGTSP1-RGTSP4	Must be set to 1
PLC parameters	K16.2	In multi-spindle control, the spindle speed S 1: Execute together with M code, 0: Specify by P	Set to 1
	K16.5	The first spindle position speed switch	Multi-spindle must be set to 1
	K16.6	The second spindle position speed switch	Multi-spindle must be set to 1
	K16.7	3rd spindle position speed switch	Multi-spindle must be set to 1

2.10 Function of spindle

2.10.1 Spindle encoder selection

The system has two encoder input interfaces (CN21, CN22). By default, the GSK-Link communication spindle feedback value is used as the feedback input of the spindle speed. In order to correctly read the actual speed of the spindle during use, it is necessary to set the relevant parameters and signals of each spindle encoder.

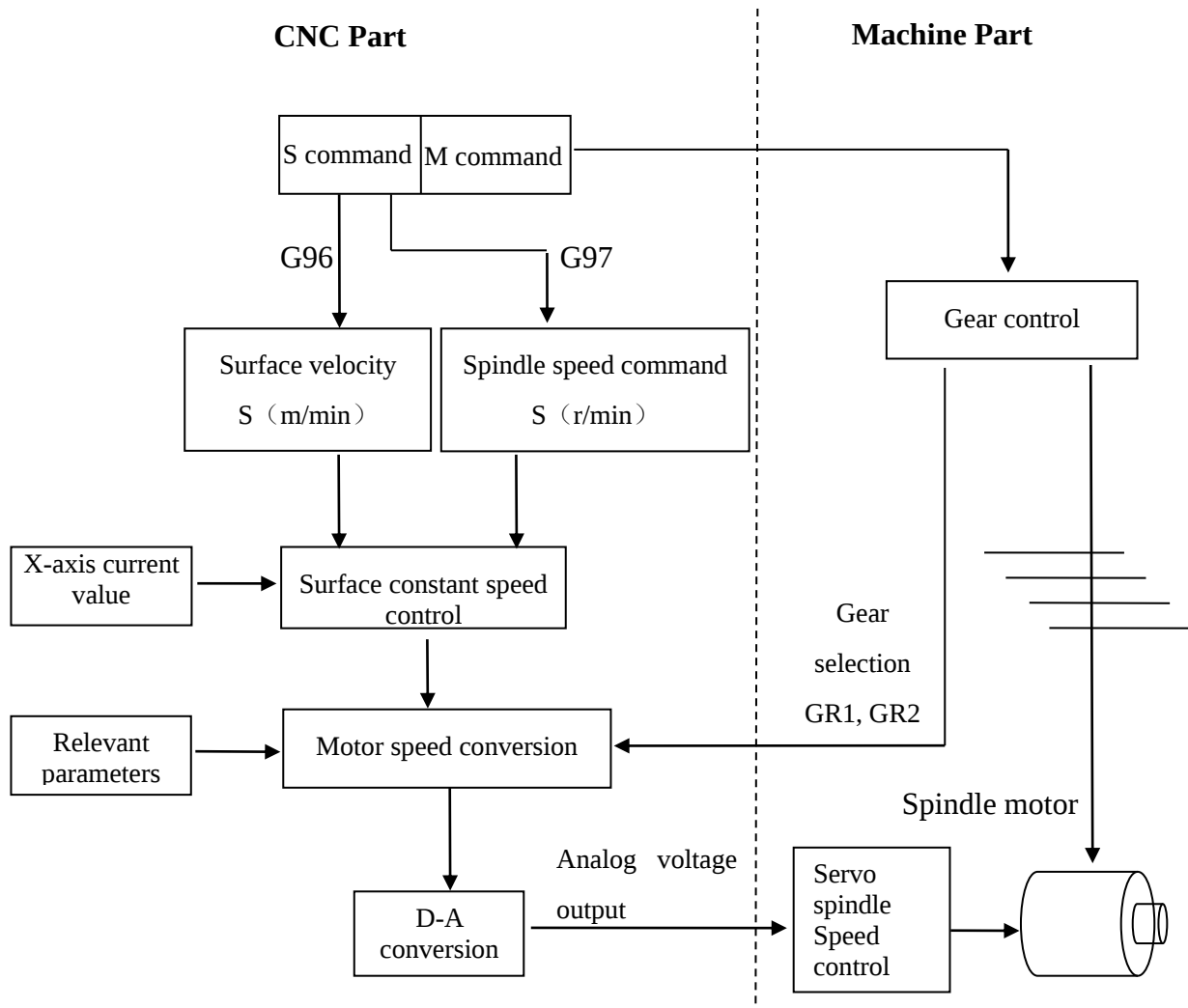
Table 2-21

Relevant Parameters			
Parameter Number	Bit	Parameter Signification	Note
3720		Number of encoder lines for each spindle	Set the number of encoder lines for each spindle (100~99999999)
3721		Number of teeth on one side gear of each spindle position encoder	These two parameters are used to set the gear ratio during speed control. (Range of setting value: 1~9999)
3722		Number of teeth on one side gear of each main axis	
3723		Channel number corresponding to each spindle encoder	This parameter sets the channel number corresponding to the spindle encoder

2.10.2 Spindle speed control

There are two ways to control the spindle speed, which can be selected by parameters. The first is to use communication to control the servo drive unit when connecting to the GSK-Link spindle. The second is to use the analog voltage output port of the extended I/O unit to output and send an analog voltage of 0V to 10V to the spindle servo drive device or inverter. Infinitely variable speed can be achieved no matter which method is adopted.

Although S command is the spindle speed, the actual control object is the spindle motor. Therefore, CNC must have a certain correspondence between the speed and grade of the spindle motor. This system determines the gear stage currently used by the machine tool by gear selection signals (GR1, GR2), and CNC outputs the spindle speed corresponding to the gear stage. The spindle speed control process is as follows:



When the programmed speed is inconsistent with the actual spindle speed, you can adjust the data parameters No. 3730 and No. 3731 to make the designated speed consistent with the actual speed. The adjustment method is divided into two methods according to whether the spindle encoder is connected or not.

Spindle encoder not used:

- ① Before adjustment, you need to set parameter 3730 to 1000 (the gain adjustment data of spindle speed analog output), and parameter 3731 to 0 (compensation value of spindle speed analog output offset voltage). After power off, disconnect the CNC and the spindle. And re-power on, execute a commonly used M code (M41-M44) of the spindle gear (the system defaults to the first gear after the system is powered on);
- ② Enter the S code of the highest speed of the gear instructed by the next command. For example, after selecting the first gear, enter the command of the highest speed of the first gear (parameter 3741) in the entry page and press the cycle start button;
- ③ Measure the output voltage SVC according to the interface definition given in the previous section;
- ④ Set the value of the following formula in parameter 3730:

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

- ⑤ After setting the parameters, again command the spindle speed (parameter 3741) of the first speed analog output of the spindle as the maximum voltage spindle speed (parameter 3741), and confirm that the output voltage should be 10V;
- ⑥ Instruct S0 command on the input page;
- ⑦ Measure the output voltage SVC;
- ⑧ Set the value of the following formula in parameter 3731:

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

- ⑨ After setting the parameters, command S0 again to confirm that the voltage is 0 V.

Use spindle encoder

- ① Before adjustment, it is necessary to set parameter 3730 to 1000 (gain adjustment data of spindle speed analog output), and parameter 3731 to 0 (compensation value of spindle speed analog output offset voltage). After the spindle encoder is correctly connected and set, execute one M codes of commonly used spindle gears (M41-M44) (the system defaults to the first gear after the system is powered on);
- ② In the input page, command the S code of the highest speed of the commanded gear. For example, after selecting the first gear, enter the first highest speed command (parameter 3741) in the input page and press the cycle start button to make the spindle rotate;
- ③ Record the actual speed value in the position page. At this time, the actual value should be similar to the specified value. If there is too much difference, please check whether the encoder parameters are set correctly;
- ④ Set the value of the following formula in parameter 3730:

$$\text{Setting value} = \frac{\text{Parameter 3741 set value}}{\text{Actual speed}} \times 1000$$

- ⑤ After setting the parameters, again command the spindle speed of the first gear speed analog output as the maximum voltage (parameter 3741), and confirm that the actual speed should be the value set in parameter 3741;
- ⑥ Instruct S0 command on the input page;
- ⑦ Record the actual speed value in the position page;
- ⑧ Enter the recorded actual speed in parameter 3731;
- ⑨ After setting the parameters, command S0 again to confirm that the output speed is 0.

Table 2-22

Relevant Parameters			
Parameter Number	Bit	Parameter Signification	Note
3031		Set the allowable digits of S code	
3708	#0	Whether to check the spindle speed arrival signal 0: Do not check 1: Check	
3708	#1	Whether to check the spindle speed arrival signal when starting thread cutting 0: Determined by parameter 3708#0 or #2 bit 1: Check	
3708	#6	In the thread processing or tapping cycle, the spindle override 0: Invalid (fixed at 100%) 1: Valid	
3710		CNC control spindle number	Set the number of spindles controlled by CNC (1~4)
3713	#6	In multi-spindle control, when the spindle is selected by a program command based on the address P, whether to automatically perform thread cutting/feed per revolution according to the selected spindle is to switch the position encoder feedback used in the etc. 0: Not to switch 1: To be switched	
3717		Amplifier number of each spindle	When the spindle is connected via GSK-Link, set to 1~99 (consistent with the spindle drive unit setting) When the spindle is connected through the remote I/O unit, set to -1~ -4 (corresponding to the remote I/O unit 1 spindle interface), -11~ -14 (corresponding to the remote I/O unit 2 spindle interface), -21~ -24 (Corresponding to the remote I/O unit 3 spindle interface), -31 ~ -34 (corresponding to the remote I/O unit 4 spindle interface)
3720		Number of encoder lines for each spindle	Set the number of encoder lines for each spindle (100~99999999)

Relevant Parameters			
Parameter Number	Bit	Parameter Signification	Note
3721		Number of teeth on one side gear of each spindle position encoder	These two parameters are used to set the gear ratio during speed control. (Range of setting value: 1~9999)
3722		Number of teeth on one side gear of each main axis	
3723		Channel number corresponding to each spindle encoder (0~2)	When set to 0, the speed is fed back through GSK-Link Set as 1~2 corresponding to encoder interface 1~2
3730		Gain adjustment data of each spindle speed analog output	Setting value range: 500~2000, data unit 0.1%
3731		Compensation value of each spindle speed analog output bias voltage	Setting value range: -1000~1000
3740		Detect the delay time of the spindle speed arrival signal	
3741		Maximum spindle speed of gear 1	Set the speed of each spindle corresponding to the gear (0 ~ 65536 r / min)
3742		Maximum spindle speed of gear 2	
3743		Maximum spindle speed of gear 3	
3744		Maximum spindle speed of gear 4	
3770		The axis used as the basis for calculation during constant line speed control	Set the axis (0, 1 to the number of controlled axes) used as the basis for calculation during constant surface speed control. When it is set to 0, the X-axis will be used as the basis
3771		Minimum spindle speed under constant surface speed control	Minimum speed when G96 is set (0~65536r/min)
3772		The upper limit speed of each spindle (0~65536r/min)	
3775		The Default Spindle Selection P Command Value in Multiple Spindles (MPD)	Set the default P command value when there is no command S_ P_ after power on

2.11 Position switch function

A total of 16 position switch functions can be set for the system control axis. When the movement distance of each axis of the control axis exceeds the distance range set by the position switch of the axis, the PLC signal output signal of the position switch corresponding to the axis is 0. When the machine coordinate of the control axis with the position switch is set When in the range, the corresponding PLC signal output is 1. The position switch function does not have a switch entity, but directly sets the number of position switches on each axis and the range of the signal sent by the switch by parameters, which is different from the general automatic control switch signal.

Parameters/Signals	Description
6910~6925	Set the serial number of the control axis that executes each position switch. When the set value is 0, it means invalid.
6930~6945	Set the maximum value of the operating range of each position switch. ISC:-99999.9999~99999.9999 ISB: -999999.999~999999.999
6950~6965	Set the minimum value of the operating range of each position switch. ISC:-99999.9999~99999.9999 ISB: -999999.999~999999.999
F70, F71	PLC signal, control axis position switch output signal PSWx

The F70 and F71 signals inform that the machine coordinates of the control axis specified by the parameters No.6910~No.6925 are within the range specified by the parameters (No.6930~No.6945, No.6950~No.6965) .

For example, set the control axis number set by parameter 6910 to 1, 6930 to 100000, and 6950 to 0. When the machine coordinate of the control axis is 3.0000mm, the F70.0 signal will be set to 1. When the machine coordinate of the control axis exceeds the parameter setting range, the F70.0 signal is set to 0.

Note 1: Set the control axis number range of the position switch from 0 to the total number of control axes.

Note 2: After the reference point return operation is completed, the position switch function is valid.

Note 3: Set the control axis number of the position switch. When it is set to 0, it means that the position switch function is not used.

2.12 Parallel program function

Parallel programs can run programs different from the main program at the same time during automatic operation. As a result, it is possible to control the loader axis and back drill axis in parallel with the main program.

The parallel program function can be triggered by the G signal to call the specified program. In addition, the parallel program can be loaded in the main program and then executed. As an auxiliary function, this function can be executed in parallel with the automatic running program.

The parallel program function currently only supports G00, G01, G02,G03,G04, G28, G35, S instructions and most M auxiliary codes.

Note 1: Before using the parallel program function, a dedicated PLC program support is required.

Note 2: Parallel program function does not support M99 instruction.

Note 3: The motion instructions of the parallel program cannot conflict with the main program being executed.

2.12.1 Parallel program editing



LOCAL

Under the program page, press the **LOCAL** button, and edit the parallel program in accordance with the common program editing method in the corresponding CNC system programming and operation



PARAM

manual. Then, after pressing **PARAM**, set the program name of the parallel program through parameters No. 3510 (PSEN1), No. 3511 (PSEN2), No. 3512 (PSEN3), and No. 3513 (PSEN4), as shown in Fig. 2-16, set the parameter No. The values of 3510 (PSEN1), No. 3511 (PSEN2), No. 3512 (PSEN3), and No. 3513 (PSEN4) are set to 1820, 1821, 1822, and 1823, respectively.

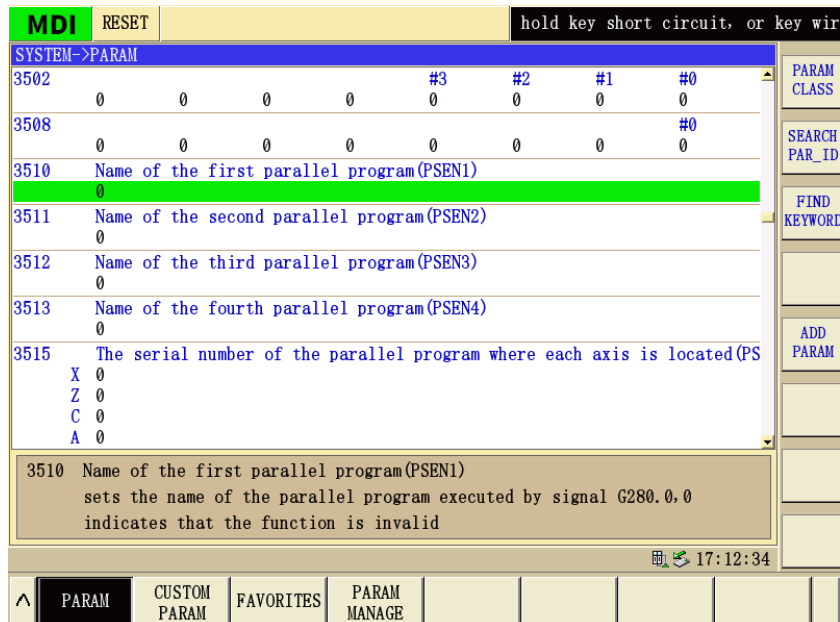


Fig. 2-16

2.12.2 Parallel program execution

There are two loading methods for the execution content of the parallel program. One is to determine the program number of the program to be executed by the parameters 3510~3513; the other is to load the parallel program in the main program. The choice of these two methods is determined by the 3500.0~3500.3 parameters.

2.12.2.1 Parallel program content is determined by parameters

When a parameter of 3500.0~3500.3 is set to 0, it means that the content of the executed parallel program is correspondingly set by No. 3510 (PSEN1), No. 3511 (PSEN2), No. 3512 (PSEN3), and No. 3513 (PSEN4). Procedure number decision. When the program number is set (as shown in Fig. 2-16), there are:

When the G280.0 signal is 1, execute the program O1820.CNC.

When the G280.1 signal is 1, execute the program O1821.CNC.

When the G280.2 signal is 1, execute the program O1822.CNC.

When the G280.3 signal is 1, execute the program O1823.CNC.

2.12.2.2 Content of parallel program is determined by main program

Another way to load the execution content of the parallel program is to load the parallel program in the main program. First, the number of bits corresponding to 3500.0~3500.3 needs to be set to 1. The loading method is as follows.

G101:	The first parallel program registration starts
.....	
.....	
G100:	The first parallel program registration ends
G102:	The second parallel program registration starts
.....	
.....	
G100:	End of the second parallel program registration
G103:	The third parallel program registration starts
.....	
.....	
G100:	The third parallel program registration ends
G104:	The fourth parallel program registration starts
.....	
.....	
G100:	The fourth parallel program registration ends

For example:

(O1139)
 G1 Z100
 G101
 G1 X100 F100
 U100
 X-100
 G100
 W-100
 Z-100
 M30

Loaded into the first parallel program, the main program skips without execution

When the main program is executed, the parallel program content set in G101~G100, G102~G100, G103~G100, G104~G100 will be stored in the memory, and the main program will not execute the parallel program content. After loading the parallel program:

When the G280.0 signal is 1, execute the first parallel program.

When the G280.1 signal is 1, execute the second parallel program.

When the G280.2 signal is 1, execute the third parallel program.

When the G280.3 signal is 1, the fourth parallel program is executed.

Note 1: The parallel program registered from the main program will be stored in the volatile memory. Therefore, it will be erased once the power is cut off.

Note 2: When the parallel program is in the running state, if the login operation of the parallel program is performed at this time, the operation is invalid.

Note 3: When the parallel program is running, if the value of parameter #3500.0-#3500.3 is modified at this time, the parameter is invalid at this time. When the parallel program is re-run, the modified parameter value is valid at this time.

Note 4: The registered program segment cannot exceed 100 segments, otherwise the alarm will be issued.

Note 5: After G101 (G102/G103/G104) is commanded, G100 must be commanded, otherwise an alarm will be issued.

Note 6: After G101 (G102/G103/G104) and G100 are commanded, if G101 (G102/G103/G104) and G100 are re-designated, it will alarm.

Note 7: G101, G102, G103, G104, and G100 must be commanded in a single block, otherwise an alarm will be issued.

Note 8: When the main program is executed in a single block, the block between G101 (G102/G103/G104) and G100 will not be executed.

Note 9: Reset/emergency stop will not erase the parallel program content in the volatile memory.

2.12.3 Parallel program display

When the corresponding program is executed manually, the corresponding serial number will be displayed on the lower right corner of the CNC interface. Fig. 2-17 shows the page status when the first program is manually executed. A green icon is displayed in the lower right corner , which means that the second manual program is being executed.



Fig. 2-17

When the program is in parallel, the icon in the lower right corner indicates the running status of the current program.

- **Automatic operation status is displayed, and the icon is green at this time:**
 - 1**: Represents the first program executed manually, that is, the G280.0 signal is 1.
 - 2**: Represents the second program executed manually, that is, the G280.1 signal is 1.
 - 3**: Represents the third program executed manually, that is, the G280.2 signal is 1.
 - 4**: Represents the fourth program executed manually, that is, the G280.3 signal is 1.
- **Pause status is displayed, and the icon is yellow:**
 - 1**: Represents the first program of manual pause, that is, the G281.0 signal is 1.
 - 2**: Represents the second program of manual pause, that is, the G281.1 signal is 1.
 - 3**: Represents the third program of manual pause, that is, the G281.2 signal is 1.
 - 4**: Represents the fourth program of manual pause, that is, the G281.3 signal is 1.
- **The program execution error status is displayed, and the icon is red at this time:**
 - 1**: It means there is an error in the manual execution of the first program.
 - 2**: It means that there is an error in the manual execution of the second program.
 - 3**: It means that there is an error in the manual execution of the third program.
 - 4**: It means there is an error in the manual execution of the fourth program.

2.12.4 Parallel program drilling, boring and rigid tapping

Drilling, boring, and rigid tapping are allowed in parallel programs. The usage method is consistent with the usage method in the main program. Please refer to the corresponding CNC system programming and operation manual for the content of drilling, boring and rigid tapping.

Parameter Number	Content
3500.0~3500.3	Contents of the 1 st ~4 th parallel programs: 0: Determined by 3510~3513 parameters 1: Determined by the main program
3501.0~3501.3	Rigid tapping methods for the first to fourth parallel programs: 0: M code for commanding rigid tapping 1: G84, G88 are used as G codes for rigid tapping
3502.0~3502.3	When the address Q is specified for rigid tapping and drilling in the 1 st to 4 th parallel programs: 0: High-speed deep hole tapping cycle 1: Deep hole tapping cycle
3508.0	Whether to display the parallel program page 0: No display 1: Display
3510~2513	Set the name of the 1 st ~4 th parallel program
3530~3533	Set the M codes for rigid tapping of the 1 st to 4 th parallel program commands.
3540~3543	Set the return amount of drilling for deep hole tapping in the first to fourth parallel programs
G280.0~G280.3	1 st ~4 th parallel program execution signal
G281.0~G281.3	1 st ~4 th parallel program pause signal
G290.0~G290.3	1 st ~4 th parallel program rigid tapping signal
G292.0~G292.3	The first parallel program rigid tapping and drilling spindle selection signal
G293.0~G293.3	The second parallel program rigid tapping and drilling spindle selection signal
G294.0~G294.3	The third parallel program rigid tapping and drilling spindle selection signal
G295.0~G295.3	The 4 th parallel program rigid tapping and drilling spindle selection signal

Note 1: To use drilling, boring or rigid tapping, it is necessary to set the spindle selection signal (G292.0~G292.3, etc.). For detailed signal introduction, please refer to Chapter 4 in the corresponding CNC system PLC manual introduction to the section "4.10.14 Parallel Programs".

Note 2: Parameters No.3530-No.3533 can be used to specify the M code for rigid tapping. When it is specified as 0, the default is M29.

Note 3: Use the designated rigid tapping method to temporarily designate S designation or axis movement command between M29 and G84/G88 blocks to generate an alarm.

Note 4: Repeatedly specify M29 to alarm during the tapping cycle.

Note 5: M29 S__ instructs the rigid tapping mode, PLC will switch accordingly after receiving M29, and the spindle will stop rotating. The spindle output in the M29 segment is equivalent to the output S0.

Note 6: When drilling, boring and rigid tapping are executed continuously in the parallel program, the next block will be executed after the tool returns to the R plane. Only when the command is G84->G01, the last tapping cycle will end. Return to the R plane.

2.12.5 Precautions for drilling/boring cycles

- 1) During reset and emergency stop, the modal of the drilling cycle is cancelled, and after the power is off, the modal of the drilling cycle is cancelled.

- 2) The Q value is specified by the radius value.
- 3) When in the drilling cycle, the T and S codes cannot be commanded, otherwise an alarm will be issued.
- 4) When in the drilling cycle, the drilling axis cannot be switched, otherwise an alarm is issued.
- 5) Except for the number of repetitions K, all other data are modal data, which will change when re-designated.
- 6) Except that the C-axis clamping M code specified by parameter #5110 can be specified during the drilling cycle, otherwise it will alarm.

2.12.6 Precautions for tapping cycle

- 1) When resetting and emergency stop, the modal of the tapping cycle is cancelled, and after the power is off, the modal of the tapping cycle is cancelled.
- 2) The Q value is specified by the radius value.
- 3) When G80 or G01/G02/G28/G35 commands are specified, the tapping cycle mode is cancelled.
- 4) If the T command is specified during tapping, it will alarm.
- 5) If the rigid tapping mode M code or S command is specified in the tapping cycle, the tapping cycle mode is canceled.
- 6) If an M command other than the C-axis clamping M code is specified during tapping, an alarm will be generated.
- 7) If the tapping axis is switched or the properties of the tapping axis are wrong during the tapping cycle, an alarm will be issued.
- 8) When No.3501.0-No.3501.3 is 1, that is, when G84/G88 is designated as the G code for rigid tapping, if there is no S code in the same segment, an alarm will be issued; when No.3501.0-No.3501.3 is 0, That is, when the M code of rigid tapping is specified, if there is no S code in the same segment, an alarm will be issued.
- 9) If the K value exceeds the maximum value of 99, an alarm will be issued. K is only valid in the block where it is designated.
- 10) If the tapping length is not commanded, or the tapping length is less than the lead, an alarm will be issued.
- 11) If the F value is not commanded, an alarm is issued.
- 12) In the tapping cycle, except for the number of repetitions K, the other tapping data are all modal data, which will be changed when re-designated.

2.13 Indexing function of indexing table

The system can index the index table by commanding the indexing position of the indexing axis (the axis must be rotated, and the parameter ROTx (No.1006#0) is set to "1"). Before and after the indexing table moves, the indexing table is automatically released or clamped.

2.13.1 Function setting

The axis name of the control axis for indexing table indexing is generally A, B or C. In the following description, the name of the control axis for indexing table indexing is set to B axis. When the actual mechanical upper axis name is not the B axis, please understand it as the axis name. The program command used for indexing table indexing uses the value following B to command the positioning angle. It must be commanded in a separate block.

Value range of indexing axis:

Indexing-axis	Metric Input (mm)	Imperial Input (inch)
	(ISB system) -99999.999 ~ 99999.999 (deg)	(ISB system) -99999.999 ~ 99999.999 (deg)
	(ISC system) -9999.9999 ~ 9999.9999 deg	(ISC system) -9999.9999 ~ 9999.9999 (deg)

Programming Example:

G0 B100; (positioned to 100°)

G0 B10000.999; (ISB incremental system. Positioning to 10000.999°)

Note: To use the indexing table function, the corresponding PLC support is required.

2.13.2 Action steps for indexing table positioning

There are two types: type A and type B, but the difference between type A and type B is that only the timing of the B-axis position control servo on/off is different, and the basic steps remain the same.

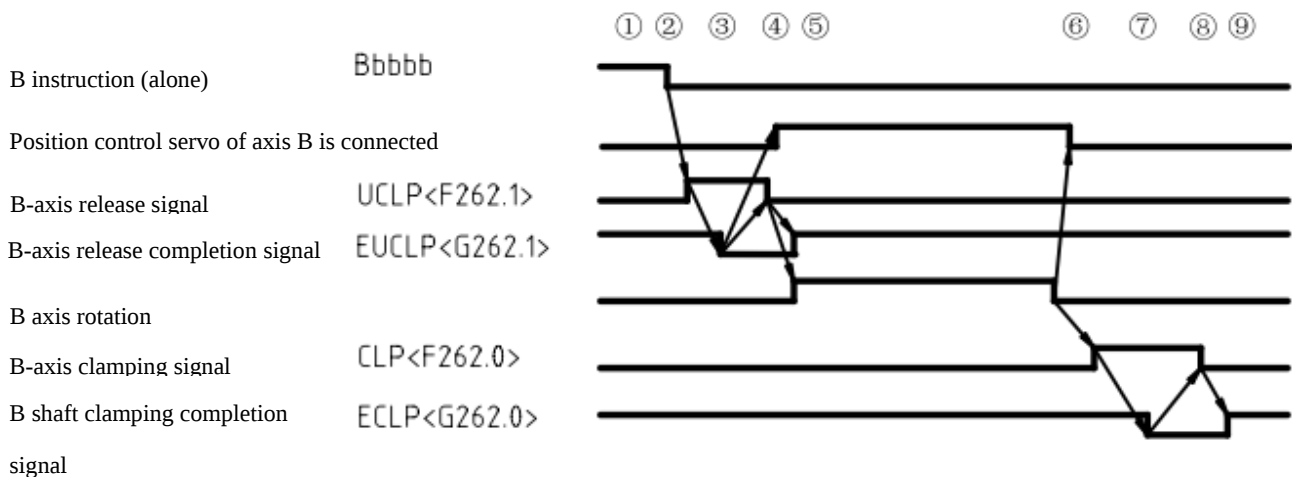
- ① Assume that Bbbbb is instructed by the instruction program.
- ② When Bbbbb is commanded, the CNC sets the B-axis release signal UCLP to '1'. In the case of type B, set the signal UCLP to '1' and at the same time set the position control of the B axis to the servo-on state.
- ③ When the UCLP signal becomes '1', the PLC side releases the clamp of the B axis. After this action is completed, the B axis release completion signal EUCLP is set to '0'.
- ④ When the B-axis release completion signal EUCLP becomes '0', the CNC sets the B-axis release signal UCLP to '0', and then informs the fact that the EUCLP signal has been received.
- ⑤ When the B-axis release signal UCLP becomes '0', the PLC side sets the B-axis release completion signal EUCLP to '1'. In the case of type A, the CNC sets the B-axis release signal UCLP to '0' and at the same time sets the B-axis position control to the servo-on state. Then, the B-axis is rotated, and the B-axis is stopped at the commanded position. The B axis always moves in rapid traverse mode.
- ⑥ When stopping the B axis at the commanded position, the CNC sets the B axis clamp signal CLP to '1'. In the case of type A, the CNC sets the signal CLP to '1' and at the same time sets the position control of

the B axis to the servo off state.

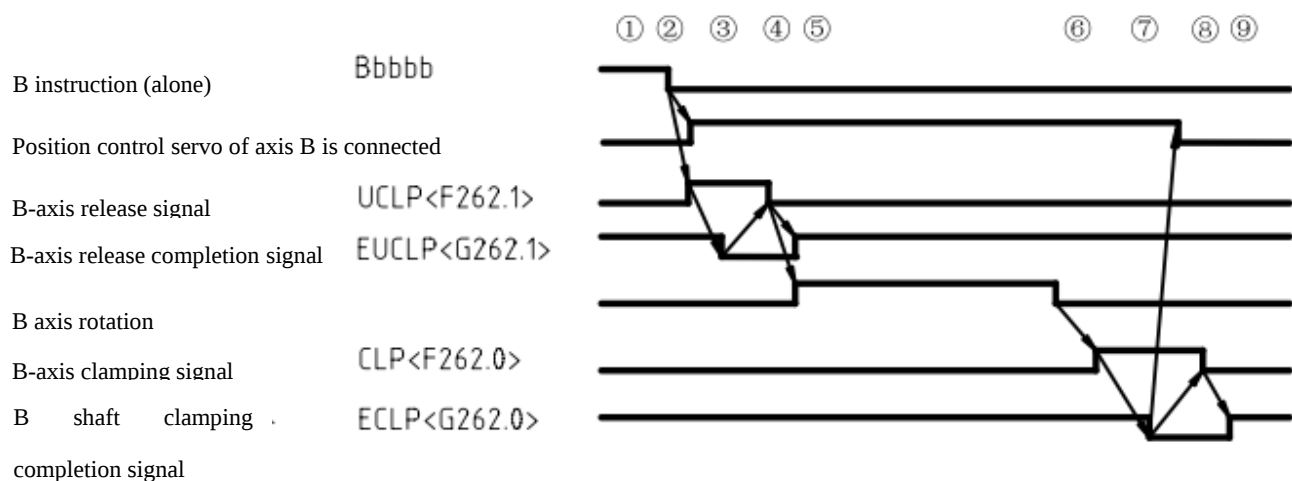
- ⑦ When the B-axis clamp signal CLP becomes '1', the PLC side mechanically clamps the B-axis with bite or injection pins. If the action of mechanically clamping the B-axis has been completed, the PLC side sets the B-axis clamping completion signal ECLP to '0'.
- ⑧ When the B-axis clamping completion signal ECLP becomes '0', the CNC sets the B-axis clamping signal CLP to '0', and then informs the fact that the ECLP signal has been received. In the case of type B, set the B-axis clamp signal CLP to '0' and at the same time set the B-axis position control to the servo off state.
- ⑨ When the B-axis clamping signal CLP becomes '0', the PLC side sets the B-axis clamping completion signal ECLP to '1'. At this point, all sequences of actions for positioning the indexing table are completed.

When the above situation is represented by a time chart, it is as shown in the Fig. below.

Timing Diagram of Type A When IDX=0:



Timing Diagram of Type B When IDX=1:



F193: Position control servo on signal

Type A, Type B test methods:

- ◆ When G262.1 signal is not given, F262.1 is 1, the position control servo ON signal of type A is 0; type B is 1;
- ◆ When G262.0 signal is not given, after running, F262.0 will be 1, the type A position control servo ON signal will be 0; the type B will be 1.

As mentioned in the basic procedure, the difference between Type A and Type B is only the timing of the B-axis position control servo on/off.

Note 1: Type A as the clamping method of B axis is suitable for clamping by injection pin. Type B as the clamping method of B axis is suitable for clamping by biting.

Note 2: The indexing axis of the indexing table always rotates in rapid movement. Dry run is invalid for the indexing axis of the indexing table.

Note 3: If resetting is performed while waiting for completion of clamping or releasing, the clamping signal or releasing signal will be cleared. In addition, the CNC exits from the state waiting for completion.

Note 4: When the movement amount is 0, the clamping/releasing operation will not be executed. In the automatic reference point return of G28, even if the movement amount is 0, the clamping/releasing operation will be executed.

Note 5: In the block containing the instruction of the indexing table axis, the acceleration/deceleration before interpolation is invalid.

Note 6: The axis used in the PLC axis control and magnetic pole position detection function cannot be controlled as an index table axis.

2.14 Additional axis function

Additional axis control refers to the extended axis used to meet the additional functions of the machine tool, such as loading and unloading mechanism, tool changing mechanism, etc. Although the PLC axis control function can meet some of the requirements, problems such as small number of control axes, difficult programming and debugging, so additional axis control functions are proposed.

The total number of control axes of the additional axis is 4, which is different from the system control axis number of 8 axes (8130 parameter setting value), which is equivalent to the system total control axis number of 12 axes.

At present, the additional axis program only supports the G commands shown in Table 2-23, and most of the M commands (for details, please refer to the description of the M command in the corresponding CNC system programming and operation manual), but does not support the T command and spindle movement.

Table 2-23 G commands supported by additional axis programs

G Code System		Function	Group	Modal
Programming Command (Set A)	Programming Command (Set B)			
*G00	*G00	Fast positioning (fast moving)	01	Modal
G01	G01	Linear interpolation		
G02	G02	Circular interpolation (clockwise)		
G03	G03	Circular interpolation (counterclockwise)		
G04	G04	Pause, orientation	00	Non-modal
G17	G17	XpYp plane selection	16	Modal
*G18	*G18	ZpXp plane selection		
G19	G19	YpZp plane selection		
G28	G28	Reference point return	00	Non-modal
G30	G30	Return to reference point 2, 3, 4		
G35	G35	Torque control		

Note 1: The function of the above command is the same as that of the conventional axis. For details, please refer to the description of the G command in the corresponding CNC system programming and operation manual

Note 2: Before using the additional axis function, special PLC program support is required.

Note 3: The additional axis program does not support the subprogram call function, including M98 subprogram call, G65 macro program call and G66 macro program modal call.

2.14.1 Basic parameter setting of additional axis

2.14.1.1 Additional axis name

The total number of control axes of the additional axis is 4 axes, and the related setting parameters of the axis name are as follows.

Relevant Parameters		
Parameter Number	Parameter Signification	Note
8410	Axis number setting for additional axis control	The parameter setting value cannot be greater than 4
8412	Set the axis name of each additional axis	The name of each additional axis set cannot be the same
8413	The attributes of each axis in the basic coordinate system of the additional axis	
8414	Station address of additional axis control (servo axis number)	

2.14.1.2 Type setting of additional axis

Set the additional axis as a linear axis or a rotary axis according to the processing requirements. The relevant parameters are as follows.

Relevant parameters		
Parameter Number	Bit	Parameter Signification
8404	#1, #0	00: Linear axis 01: Rotary axis (Type A) 11: Rotary axis (Type B)

2.14.2 Calculation and setting of additional axis gear ratio

The calculation of the gear ratio of the additional axis is the same as that of the system control axis. For details, please refer to "2.4 Gear Ratio Calculation and Setting" in this chapter. The related parameters of additional axis gear ratio setting are shown in the table below.

Parameter Number	Parameter Signification
8417	Set the command multiplication ratio of each additional axis (the numerator of the gear ratio)
8418	Set the detection multiplication ratio of each additional axis (gear ratio denominator)

2.14.3 Servo parameter setting of additional axis

After the system and the servo are correctly connected and powered on, correctly set the encoder type, the direction of pulse output of each additional axis and the movement direction of each additional axis, etc. The specific setting steps are as follows:

- ① According to the encoder type of the servo motor, set parameter No.8408#5 (AAPCx) correctly to select whether the servo uses an absolute encoder.
- ② Set the gear ratio CMR/DMR (parameter No.8417 and No.8418) correctly according to the transmission ratio of the machine tool, so that the movement distance of the machine tool is consistent with the program command value.
- ③ If the movement direction of the machine tool is inconsistent with the direction required by the displacement command, parameter No.8407#2 (APODx) can be set to reverse the movement direction of the corresponding servo axis command output.

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
8407	#0	Each additional axis removal function 0: Valid 1: Invalid
8407	#2	Set the pulse output direction of each additional axis 0: No negation 1: Negation
8408	#5	Set the position detector of each additional axis 0: Do not use absolute encoder 1: Use absolute encoder
8426		Set the backlash compensation amount of each additional axis
8427		Set the backlash compensation frequency value of each additional axis

2.14.4 Additional axis acceleration/deceleration feature adjustment

The larger the acceleration/deceleration time constant, the slower the acceleration/ deceleration process, the smaller the impact of the machine tool movement, and the lower the processing efficiency; the smaller the acceleration/deceleration time constant, the faster the acceleration/ deceleration process, and the greater the impact of the machine tool movement, and the higher the efficiency at the time.

When the acceleration/deceleration time constant is the same, the higher the start/end speed of acceleration/deceleration, the faster the acceleration/deceleration process, the greater the impact of the machine tool movement, the higher the processing efficiency; the lower the start/end speed of acceleration and deceleration, the slower the acceleration/deceleration process, the smaller the impact of the machine tool movement, and the lower the processing efficiency.

The principle of acceleration/deceleration adjustment is to appropriately reduce the acceleration /deceleration time constant and increase the start/stop speed of acceleration and deceleration on the premise that the drive unit does not alarm, the motor does not lose step and the machine tool movement has no obvious impact, so as to improve the processing efficiency. If the acceleration/deceleration time constant is set too small, and the acceleration/deceleration start/end speed is set too high, it is easy to cause the drive unit to alarm, the motor is out of step, or the machine tool vibration.

Setting of related parameters in the following situations:

- **Fast moving**

Parameter Number	Parameter Signification
8450	Set the rapid traverse speed of each additional axis when the rapid override is 100%
8451	Set the speed of each additional axis when the rapid traverse override is F0
8460	Set the acceleration and deceleration time constant of each additional axis rapid traverse

- **Manual MPG feed**

Parameter Number	Parameter Signification
8452	Set the speed of each additional axis during manual continuous feed
8453	Set the maximum feed rate of each additional axis manual MPG feed
8462	Set the acceleration/deceleration time constant of each additional axis manual feed
8463	Set the acceleration/deceleration time constant of each additional axis MPG feed

- **Cutting infeed**

Parameter Number	Parameter Signification
8459	Set the maximum speed of each additional axis during cutting feed
8464	Set the acceleration and deceleration time constant of the cutting feed of each additional axis

- **Reference point return**

Parameter Number	Parameter Signification
8456	Set the reference point return speed of each additional axis
8457	Set the speed of each additional axis after deceleration when returning to the reference point
8434	Set the X address of the deceleration signal allocated to the reference point return operation for each additional axis
8435	Set the bit address of the X address of the deceleration signal assigned to each additional axis for reference point return operation

2.14.5 Additional axis reference point and soft limit

2.14.5.1 Additional axis reference point setting

The system supports three mechanical zero point establishment (also called reference point setting) methods, non-block type reference point setting, block type reference point setting and absolute encoder reference point setting.

Reference Point Setting Mode	Relevant Parameters
Without stopper	Parameter No.8408#5 (AAPCx) is set to 0, and parameter No.8403#7 (ADJZx) Set to 1
With stopper	Parameter No.8408#5 (AAPCx) is set to 0, and parameter No.8403#7 (ADJZx) Set to 0
Absolute encoder	Parameter No.8408#5 (AAPCx) is set to 1

The reference point setting of the additional axis is roughly the same as the reference point setting of the ordinary axis, except the difference in the relevant parameters. For the detailed operation of the additional axis reference point setting, please refer to the corresponding CNC system installation and debugging manual for the axis reference point. And the description part with the soft limit. The related parameters of the additional axis reference point setting are shown in the table below.

Relevant Parameters		
Parameter Number	Bit	Parameter Signification
8400	#0	Whether the coordinate system is memorized when each axis is powered on when the additional axis is equipped with an incremental encoder 0: No memory 1: Memory
8403	#3	Deceleration signal for reference point return of each additional axis 0: Decelerate when the signal is 0 1: Decelerate when the signal is 1
8403	#4	Reference point return direction of each additional axis 0: Forward direction 1: Reverse direction
8403	#5	When the reference point of each additional axis is established and the reference point is manually returned again 0: Use the deceleration block 1: Have nothing to do with the deceleration block
8403	#7	Setting of zero return without stop in the control of each additional axis 0: Invalid 1: Valid
8408	#4	In the case of the absolute position detector in the additional axis control, the mechanical position and the absolute position detector position 0: Inconsistent 1: Consistent
8408	#5	Set the position detector of each axis in the additional axis control 0: Not used 1: Used
8419		When the additional axis is equipped with an absolute encoder, power on to establish the allowable deviation value of the machine coordinate

● **Overtravel alarm release:**

When the tool exceeds the stored stroke limit, an alarm is displayed and the tool decelerates and stops. Switch to manual mode to move the tool out of the forbidden area in the reverse direction (such as positive overtravel, move out in negative direction; if negative overtravel, move out in positive direction), press the reset button to clear the alarm display.

2.14.6 Additional axis position switch function

A total of eight position switch functions can be set for the additional axis. When the movement distance of each axis of the additional axis exceeds the distance range set by the position switch of the axis, the output signal of the position switch corresponding to this axis is 0. When the machine coordinate of the control axis with the position switch set does not exceed the set range, the corresponding PLC signal output is 1. The position switch function is different from the switch signal in the usual sense of automatic control. It does not have a switch entity but directly sets the number of position switches on each axis and the range of the switch sending signal by parameters.

Parameter Number /Signals	Description
8470~8477	Set the additional control axis number for executing each position switch. When the set value is 0, it means invalid.
8478~8485	Set the maximum value of the operating range of each position switch. (-99999.9999~99999.9999)
8486~8493	Set the minimum value of the operating range of each position switch. (-99999.9999~99999.9999)
F303	PLC signal, additional axis position switch output signal APSW

The F303 signal informs that the machine coordinates of the control axis specified by the parameters No. 8470 ~ No. 8477 are within the range specified by the parameters (No. 8478 ~ No. 8485, No. 8486 ~ No. 8493).

For example, set the control axis number of parameter No. 8470 to 1, 8478 to 100000, and 8486 to 0. When the machine coordinate of the additional axis is 3.0000mm, the F303.0 signal will be set to 1. When the machine coordinate of the additional axis exceeds the parameter setting range, the F303.0 signal is set to 0.

Note 1: The control axis number range of the set position switch is 0 to the total number of additional axis control axes.

Note 2: After the reference point return operation is completed, the position switch function is valid.

Note 3: Set the control axis number of the position switch. When it is set to 0, it means that the position switch function is not used.

Chapter 3 Network Communication Connection

The application of network in industrial automation makes production management develop towards automation and intelligence. Through the network, the automatic collection, analysis and management of numerical control system data can be realized. The system supports remote monitoring function, remote data communication function and MODBUS protocol data communication function.

3.1 Network communication setting

The network communication connection needs to set the CNC IP address and other network parameters, and need to set the external access IP address, and grant the external IP address authority, as shown in Fig. 3-1.

No.	IP allowance	Operation
1	0.0.0.0	R.
2	0.0.0.0	R.
3	0.0.0.0	R.
4	0.0.0.0	R.
5	0.0.0.0	R.

Fig. 3-1

3.2 FTP communication setting

Before enabling the FTP file transfer function, you need to set the IP address, port, account and password of the FTP server to be connected on the system, as shown in Fig. 3-2.

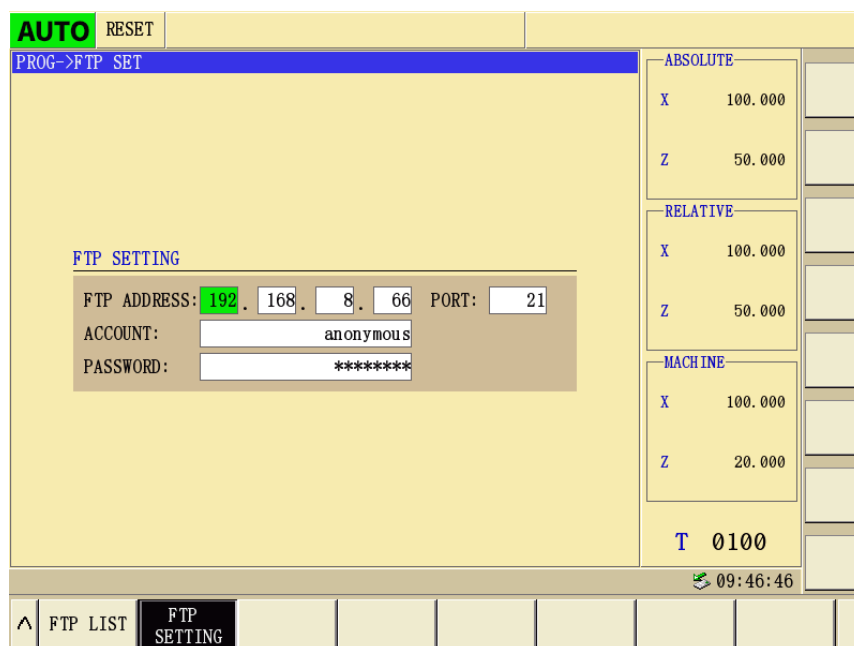


Fig. 3-2

3.3 Remote data communication function

The remote data communication function is a secondary development of the CNC system. When customers need application software on a PC to connect to the CNC system via a network for online diagnosis, production monitoring, or management, the company provides customers with the required communication protocol connection package. Customers can load this data communication protocol to establish network connections with the CNC system, including connecting a single system or multiple systems simultaneously to form a network. For any information required from the system, customers can call various communication interfaces provided by the system manufacturer, such as coordinate values, number of machined workpieces, alarm information, and various machining data.

The system must enable the remote data communication function to allow external connections. Set parameter No.930.1 (NDSVR) to "1" and configure the IP address of the PC to be connected in the IP settings page.

3.4 MODBUS data communication function

CNC system provides standard MODBUS data communication protocol, which can read most of the system information. Like the remote data communication function, this function is also a secondary development of the CNC system and is provided to customers who need to use this protocol to connect to the CNC system.

The CNC system needs to turn on the MODBUS communication switch to allow external connections. Set parameter No.930.2 (MODBUS) to "1", and set the IP address of the PC to be connected in the IP setting page.

Appendix

Appendix I Parameter Description

This chapter mainly explains the meaning of the system parameters. Different functional requirements can be achieved through different parameter settings.

There are six main types of parameter data:

Data Format		Range
(1) Bit type		8 bits 0 or 1
(2) Bit axis type		
(3) Bit spindle type		
(4) Font type		Different parameters have different set value ranges. For details, see 「Value range」 of the parameter.
(5) Word axis type		
(6) Font spindle type		

Each parameter should contain the following information:

「Modification authority」: System (level 1), machine tool (level 2), equipment (level 3), operation (level 4), restricted (level 5).

「Parameter type」: Bit type, bit axis type, bit spindle type, font type, font axis type, font spindle type

「Valid mode」 : Immediate, or power on

「Value range」: Interval, enumeration, or special judgment

「Factory default」: 8-bit binary, or 32-bit integer value

「Input and output」: The parameter may be related or irrelevant to the "input unit" and "output unit".

Note 1: The 「value range」 of bit type is 0 or 1.

Note 2: If the 「Valid mode」 is not marked, the parameter is effective immediately.

1.1 Parameters related to system settings

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Factory default」: 0000 0000

「Valid mode」: Immediate

#2 INI Input Unit

0: Metric

1: Imperial

#5 SEQ Whether to Automatically Insert the Sequence Number

0: Do not proceed

1: Proceed

Note: When programming in edit mode and input mode, the sequence number can be inserted automatically. The incremental value of the sequence number is set in parameter No.3216.

1.2 Parameters related to input and output interfaces

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Factory default」: 0000 0000

「Valid mode」: Immediate

#6 OWN When inputting or outputting NC data or programs, if a file with the same name is detected

0: Display information about whether to overwrite

1: Overwrite all files without displaying a prompt

0929	Specification Of Machining Program File Attributes (FOUT)
------	---

「Parameter type」: Font type

「Modify authority」: Equipment

「Factory default」: 0

「Valid mode」: Immediate

Set the suffix for the output machining program file name:

0: CNC

1: TXT

2: NC

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Valid mode」: Power on

「Factory default」: 0000 0000

#0 RMEN Whether to Use the Remote Monitoring Function

0: Not used

1: Used

#2 MODBUS Whether to Turn on Modbus Communication

0: Off

1: On

#3 LABTUEN Whether to Enable Remote Servo Data Sampling Function

0: Off

1: On

#7 IPC Whether to enable the IP filtering function

0: Off

1: On

	#7	#6	#5	#4	#3	#2	#1	#0
0931		BRD	MARK					

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Valid mode」: Power on

「Factory default」: 0000 0000

#5 MARK Whether to enable the TCP marking machine function

0: Off

1: On

#6 BRD Whether to enable the Modbus TCP protocol voice broadcast function

0: Off

1: On

1.3 Parameters related to axis control/setting unit

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Valid mode」: Power on

「Factory default」: 0000 0000

#0 INM The Minimum Movement Unit of the Linear Axis is

0: Metric (Metric machine tool)

1: Imperial (Imperial machine tool)

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Factory default」: 0000 0000

「Valid mode」: Immediate

#1 DLZ Is the Non-stop Reference Point Setting Function Valid?

0: Invalid

1: Valid (valid for all axes)

Note: When parameter DLZ is 0, parameter 1005#1 (DLZx) can be used to set the validity/invalidity of each axis.

#3 AZR G28 Command When the Reference Point is not Established

0: Use the deceleration block to return to the reference point

1: Alarm

Note: When using the non-block reference point setting function (parameter 1002#1 (DLZ) is "1" or parameter 1005#1 (DLZx) is "1", it has nothing to do with the setting of AZR.

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

「Parameter type」: Bit type

「Modification authority」 : Machine tool

「Valid mode」 : Power on

「Factory default」 : 0000 0000

#1 ISC Set the Minimum Input Unit and Minimum Command Increment

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

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

#3 SCC When ISC is 1, the minimum input unit and minimum command increment displayed

0: 0.0001mm or 0.00001inch

1: 0.001mm or 0.0001inch

#6 RPR Set Minimum Command Increment of Rotary Axis and Multiple of ISC Parameter

#7 IPC Set the minimum command increment of the rotary axis and the multiple of the ISC parameter

00: ×1 times

01: ×10 times

10: ×100 times

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

「Parameter type」 : Bit axis type

「Modify authority」 : Equipment

「Factory default」 : 0000 1000

「Valid mode」 : Immediate

#0 ZRNx When the reference point is not established, in the automatic operation (automatic, DNC or input), when a movement command other than G28 is specified, will the system give an alarm

0: Alarm

1: No alarm

#1 DLZx Whether the function of setting the reference point without block valid

0: Invalid

1: Valid

Note: It is valid when parameter 1002#1 (DLZ) is "0". When parameter 1002#1 (DLZ) is "1", it has nothing to do with this parameter, and the function of setting the reference point without block is valid for all axes.

#3 HJZx When the reference point has been established and then the manual reference point return is performed

0: Use the deceleration block to return to the reference point

1: It has nothing to do with the deceleration block, quickly locate to the reference point

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

「Parameter type」: Bit axis type

「Modification authority」: Machine tool

「Valid mode」: Power on

「Factory default」: 0000 0000

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

ROSx	ROTx	Content
0	0	Linear axis It can perform metric/imperial conversion. All coordinate values are linear axis type. The stored pitch error compensation is linear axis type.
0	1	Rotary axis (Type A) It cannot perform metric/imperial conversion. The machine coordinate values are displayed cyclically according to the setting of parameter 1260. The relative coordinate value is related to the parameter 1008#2, 1008#0, and the absolute coordinate is related to the parameter 1008#0. The stored pitch error compensation is a rotary axis type. Automatic reference point return (G28, G30) from the direction of returning to the reference point, the movement amount does not exceed one revolution.
1	0	The setting is invalid.
1	1	Rotary axis (Type B) It cannot perform metric/imperial conversion. Machine coordinate value, relative coordinate value (related to parameter 1008#2), absolute coordinate value is linear axis type (cannot be displayed cyclically according to parameter 1260). The stored pitch error compensation is linear axis type.

#3 DIAx Set the movement amount of each axis as

0: Radius specification

1: Diameter specification

#5 ZMIx Set the direction of each axis returning to the reference point

0: Positive direction

1: Negative direction

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

「Parameter type」: Bit axis type

「Modify authority」: Equipment

「Factory default」: 0000 0000

「Valid mode」: Immediate

#7 RZDx Rotary axis (type a) in the state of establishing the reference point, whether to select the nearest direction when the reference point is returned

0: Invalid

1: Valid

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

「Parameter type」: Bit axis type

「Modify authority」: Equipment

「Valid mode」: Power on

「Factory default」: 0000 0000

#0 ROAx Set whether the circular display function of the rotary axis is valid

0: Invalid

1: Valid

Note: ROAx is only valid for the rotation axis (parameter 1006#0 (ROT_x) is 1).

#1 RABx Set the axis rotation direction when absolute command

0: Rotation direction closer to the target

1: The direction specified by the command value symbol

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

1010	CNC Control Axis (CCA)
-------------	-------------------------------

「Parameter type」: Font type

「Modification authority」: Machine tool

「Valid mode」: Power on

「Value range」: 1~total number of control axes

Set the maximum number of axes that can be directly controlled by CNC (1~total number of controlled axes), and the rest are controlled by PLC.

Note: The total number of control axes depends on parameter No. 8130, and the setting value of this parameter cannot be greater than the value set by parameter No. 8130.

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Factory default」: 0000 0000

「Valid mode」: Immediate

#6 WIC Workpiece Origin Offset Measurement Value Directly Input

0: Only valid for the selected workpiece coordinate system

1: Valid for all coordinate systems

1020	Programming Name of Each Axis (CAN)
-------------	--

「Parameter type」: Word axis type

「Modification authority」: Machine tool

「Valid mode」: Immediate

「Value range」: 88 (X), 89 (Y), 90 (Z), 65 (A), 66 (B), 67 (C), 85 (U), 86 (V), 87 (W)

Set the axis name of each control axis.

Note: The same axis name cannot be set, the U, V, and W axes are only valid in the B set of G codes.

1022	Attributes of Each Axis in the Basic Coordinate System
-------------	---

「Parameter type」: Word axis type

「Modification authority」: Machine tool

「Valid mode」: Power on

「Value range」: 0~7

In order to determine the plane of circular interpolation, tool offset and tool nose radius:

G17: X-Y plane;

G18: Z-X plane;

G19: Y-Z plane.

Each control axis is divided into four categories: 1) basic X-axis and its parallel axis, 2) basic Y-axis and its parallel axis, 3) basic Z-axis and its parallel axis, 4) rotary axis. But each axis of the basic 3 axis can only be set one: X, Y, Z; parallel axis can be set more than 2 (parallel to the basic axis).

Setting Value	Significance
0	Neither the basic 3-axis nor its parallel axis
1	X-axis in basic 3 axis
2	Y-axis in basic 3 axis
3	Z-axis in basic 3 axis
5	Parallel to X-axis
6	Parallel to Y-axis
7	Parallel axis to Z-axis

1023

Station Address Number of Each Axis (NSA)

「Parameter type」: Word axis type

「Modification authority」: Machine tool

「Valid mode」: Power on

「Value range」: 0~99, -31, -21, -11, -1

Set the Station Address of the Feed Servo Slave Station(Range: -31, -21, -11, -1, 0~99; 0 indicates no such slave station, negative values indicate selecting the pulse interface of the I/O unit)

1025

The 2nd Programming Name of Each Axis (CAN2)

「Parameter type」: Word axis type

「Modification authority」: Machine tool

「Valid mode」: Immediate

「Value Range」: 0~9

Set the second axis name (0~9) of each control axis, 0 means invalid.

1.4 Parameters related to torque control

	#7	#6	#5	#4	#3	#2	#1	#0
1101						TJE		CTD

「Parameter type」: Bit type

「Modify authority」: Equipment

「Factory default」: 0000 0000

「Valid mode」: Immediate

#0 CTD Reset whether to clear the torque control output value

0: Controlled by G signal

1: Clear

#2 TJE Is the torque control override valid in manual mode

0: Invalid

1: Valid

1110

Whether the Torque Control of Each Axis is Valid (NCT)

「Parameter type」: Word axis type

「Modification authority」: Machine tool

「Valid mode」: Power on

「Value range」: 0~1

Set whether the torque control of each axis is valid (0~1: 0 means invalid, 1 means valid).

1111

Detection Time of Each Axis Arrival Signal When G35 is Commanded

「Parameter type」: Word axis type

「Modify authority」: Equipment

「Valid mode」: Immediate

「Value range」: 200~2000

Set the detection time (200~2000) ms of the arrival signal of each axis of G35 command.

1113

Speed Command Numerator (NCM) for Torque Control of Each Axis

1114

Speed Command Denominator for Torque Control of Each Axis (NDM)

「Parameter type」: Word axis type

「Modify authority」: Equipment

「Parameter default」: 1

「Valid mode」: Immediate

「Value range」: 1~99999

Set the speed command numerator and denominator (1~99999) of the torque control of each axis.

1120

Command Speed of Torque Control of Each Axis in Manual Mode (HNCS)

「Parameter type」: Word axis type

「Modify authority」: Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 6~6000

Set the command speed (6~6000) in manual mode during torque control of each axis.

1121

**Command Torque of Each Axis Torque Control in Manual Mode
(HNCN)**

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 400

「Valid mode」 : Immediate

「Value range」 : 10~30000

Set the command torque (10~30000)*0.01 (N·m) in manual mode during torque control of each axis.

1125

Initial Torque for Axis Torque Control in Manual Mode (ATID)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : -30000~30000

Set the Initial Torque After Each Axis Switches to Torque Control(Range: (-30000~30000) × 0.01 (N · m))

1130

**Speed Acceleration and Deceleration Time Constant under Torque
Control of Each Axis (TSPT)**

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~2000

Set the acceleration and deceleration time (0~2000) ms to reach the command speed under the torque control of each axis.

1140

D-address Corresponding to Each Axis Coordinate (TCAD)

「Parameter type」 : Word axis type

「Modify authority」: Machine tool

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~999

1160

Servo Current Detection Value of Each Axis (CEC)

「Parameter type」: Word axis type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~3000

Set the threshold of servo current detection for each axis (0~3000)*0.1A, when the set value is 0, it means no detection.

1.5 Parameters about the coordinate system

	#7	#6	#5	#4	#3	#2	#1	#0
1201	WZR		EWZ	RWO	ZCR	ZCL		

「Parameter type」: Bit type

「Modify authority」: Equipment

「Factory default」: 0000 0000

#2 ZCL After the manual reference point return is completed, the local coordinate system

0: Do not cancel

1: Cancel

#3 ZCRA After the manual reference point return is completed, the offset of the workpiece coordinate system set by g50

0: Do not cancel

1: Cancel

#4 RWO Aorkpiece coordinate system offset set by g50 when power-on coordinate memory

0: Clear

1: Restore to the memorized value at the last power-off

#5 EWZ Workpiece coordinate system when power-on coordinate memory

0: Do not return to G54

1: Return to G54

#7 WZR Workpiece coordinate system when reset

0: Do not return to G54

1: Return to G54

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Factory default」: 0000 0000

#0 EWD The moving direction of the coordinate system caused by the offset of the external workpiece origin

0: Same as the direction specified by the offset of the external workpiece origin

1: It is opposite to the direction specified by the offset of the external workpiece origin

#2 G50 When the coordinate system setting g50 code is commanded

0: Do not alarm and execute G50

1: Alarm

#3 RLC Local coordinate system of after reset

0: Do not cancel

1: Cancel

#7 FWT G50 Command execution mode

0: Mode A

1: Mode B

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Factory default」: 0000 0000

#0 MCE Whether to memorize coordinate system after power-on when equipped with incremental encoder

0: Do not memorize

1: Memorize

1206	Allowable Deviation Value of Machine Coordinate System (MER) is Established When the Absolute Encoder is Powered on
------	---

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」 : 1.0

「Valid mode」 : Immediate

「Input and output」 : Output related

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	0~9.999	0~9.9999	mm
Imperial Machine	0~9.9999	0~9.99999	inch
Rotary axis	0~9.999	0~9.9999	deg

It is used to detect the deviation alarm when the machine tool coordinate system is established after power-on. When set to 0, the deviation will not be detected.

1207

Deviation Compensation Range of Machine Coordinate System (PMCD) Is Established When Absolute Encoder Is Powered on

「Parameter type」 : Word axis type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Input and output」 : Output related

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	0~1.000	0~0.1000	mm
Imperial machine	0~1.0000	0~0.10000	inch
Rotary axis	0~1.000	0~0.1000	deg

It is used for the deviation compensation range value when the machine tool coordinate system is established after power-on. When set to 0, the deviation will not be compensated (0~1.0000).

1220

Offset Amount of External Origin of Workpiece Coordinate System of Each Axis (EWO)

「Parameter type」 : Word axis type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Input/Output」 : Input related

「Value range」 :

Value Range	IS-B	IS-C	Unit
Linear axis (metric input)	-999999.999~999999.999	-99999.9999~99999.9999	mm
Linear axis (imperial input)	-99999.9999~99999.9999	-9999.99999~9999.99999	inch
Rotary axis	-999999.999~999999.999	-99999.9999~99999.9999	deg

This is a parameter to determine the origin of the workpiece coordinate system (G54~G59). This parameter is the effective common offset for all workpiece coordinate systems.

1221	Offset Amount of Each Axis Origin of G54 Workpiece Coordinate System (WO1)
1222	Offset Amount of Each Axis Origin of G55 Workpiece Coordinate System (WO2)
1223	Offset Amount of Each Axis Origin of G56 Workpiece Coordinate System (WO3)
1224	Offset Amount of Each Axis Origin of G57 Workpiece Coordinate System (WO4)
1225	Offset Amount of Each Axis Origin of G58 Workpiece Coordinate System (WO5)
1226	Offset Amount of Each Axis Origin of G59 Workpiece Coordinate System (WO6)

「Parameter type」 : Word axis type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Input/Output」 : Input related

「Value range」 :

Value Range	IS-B	IS-C	Unit
Linear axis (metric input)	-999999.999~999999.999	-99999.9999~99999.9999	mm
Linear axis (imperial input)	-99999.9999~99999.9999	-9999.99999~9999.99999	inch
Rotary axis	-999999.999~999999.999	-99999.9999~99999.9999	deg

This is a parameter to determine the origin of the workpiece coordinate system (G54~G59). This parameter is the effective common offset for all workpiece coordinate systems.

1240	Machine Coordinate Value of Each Axis of The 1 st Reference Point (RF1)
1241	Machine Coordinate Value of Each Axis of The 2 nd Reference Point (RF2)
1242	Machine Coordinate Value of Each Axis of The 3 rd Reference Point (RF3)
1243	Machine Coordinate Value of Each Axis of The 4 th Reference Point (RF4)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial Machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch
Rotary axis	-999999.999~999999.999	-99999.9999~99999.9999	deg

Set the coordinate values of the first to fourth reference points in the machine coordinate system.

1260	Movement Amount Per Revolution of Each Axis When Rotating the Axis (PRA)
------	--

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 360.0000

「Valid mode」: Power on

「Value range」:

Value Range	
IS-B	1.000~999999.999
IS-C	1.0000~99999.9999

Set the amount of movement of the rotation axis per revolution.

1.6 Parameters related to stroke detection

The setting unit of stroke parameter 1320~parameter 1327 is as follows.

Setting Unit	IS-B	IS-C	Unit
Metric machine	0.001	0.0001	mm
Imperial 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

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Valid mode」 : Power on

「Factory default」 : 1000 0000

#0 OUT The forbidden area of stored stroke detection 2 set by parameter 1322 and Parameter 1323 is

0: Inside area

1: Outer area

#2 LMS Stored stroke detection switching signal EXLM is valid

0: Invalid

1: Valid

Note: The stored stroke detection 1 has 2 groups of parameters for setting the prohibited area.

With the stored stroke limit switching signal, the set prohibited area can be selected.

(1) Prohibited area I: parameter 1320, parameter 1321

(2) Prohibited area II: parameter 1326, parameter 1327

#5 RL3 Stroke detection 3 release signal RLSOT3 is valid

0: Invalid

1: Valid

#6 LZR After the power is turned on and before the manual reference point return, whether to perform the first stored stroke detection

0: Proceed

1: Not proceed

Note: In the case of using an absolute position encoder, the reference point has been established when the power is turned on, so it has nothing to do with this setting. After turning on the power, the stored stroke detection is directly performed.

#7 BFA When storage stroke detection 1 alarm occurs

0: The tool stops after entering the prohibited area

1: The tool stops before the prohibited area

	#7	#6	#5	#4	#3	#2	#1	#0
1301	PLC	OTS		OF1				

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00010000

「Valid mode」: Immediate

#4 OF1 When the axis moves to the movable range after an alarm occurs in the stored stroke detection 1

0: Do not clear the alarm before resetting

1: Disable OT alarm immediately

#6 OTS Whether to output an overtravel alarm signal to plc when a stored stroke detection alarm occurs

0: No output

1: Output

#7 PLC Whether to perform travel limit detection before moving 1

0: Do not proceed

1: Proceed

	#7	#6	#5	#4	#3	#2	#1	#0
1302	LIM							

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Power on

#7 LIM Whether the tool offset limit function is valid

0: Invalid

1: Valid

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

「Parameter type」: Bit axis type

「Modify authority」: Equipment

「Factory default」 : 0000 0000

#0 OT2X Whether each axis is checked for stored stroke detection 2

0: Do not proceed

1: Proceed

#1 OT3X Whether each axis is checked for stored stroke detection 3

0: Do not proceed

1: Proceed

1320	Coordinate Value of The Positive Direction Boundary of Each Axis Stored Stroke Detection (PC1)
1321	Coordinate Value of The Negative Direction Boundary of Each Axis Stored Stroke Detection 1 (NC1)

「Parameter type」 : Word axis type-real number type

「Modify authority」 : Equipment

「Valid mode」 : Immediate

「Input and output」 : Output related

「Factory default」 : NO.1320 is 999.9999, NO.1321 is -999.9999

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch
Rotary axis	-999999.999~999999.999	-99999.9999~99999.9999	deg

Set the boundary coordinate values in the machine coordinate system of the positive and negative directions of the stored stroke detection 1 of each axis respectively. The outside of the set boundary is the forbidden tool entry area.

Note 1: The axis specified by the diameter is set with the diameter value.

Note 2: When setting (parameter 1320) < (parameter 1321), the stroke is infinite, and the stored stroke 1 check cannot be performed. (The stored stroke limit switch signal is invalid). If an absolute command is specified, the coordinate value may overflow and normal movement cannot be performed.

1322	Coordinate Value of The Positive Direction Boundary of Each Axis Stored Stroke Detection 2 (PC2)
1323	Coordinate Value of The Positive Direction Boundary of Each Axis Stored Stroke Detection 2 (NC2)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch
Rotary axis	-999999.999~999999.999	-99999.9999~99999.9999	deg

Set the coordinate values of the positive and negative direction boundaries of each axis's memory-type stroke detection 2 in the mechanical coordinate system respectively. Whether the outside or inside of the set boundary is the prohibited area is determined by parameter 1300#0 (OUT).

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

1324	Coordinate Value of The Positive Direction Boundary of Each Axis Stored Stroke Detection 3 (PC3)
1325	Coordinate Value of The Positive Direction Boundary of Each Axis Stored Stroke Detection 3 (NC3)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch
Rotary axis	-999999.999~999999.999	-99999.9999~99999.9999	deg

Set the coordinate values of the positive and negative direction boundaries of each axis's memory-type stroke detection 3 in the mechanical coordinate system respectively. The inside of the set boundary is the area where tool entry is prohibited.

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

1326	Coordinate Value II of The Positive Direction Boundary of Each Axis Stored Stroke Detection (PC12)
1327	Coordinate Value II of The Negative Direction Boundary of Each Axis Stored Stroke Detection (NC12)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch
Rotary axis	-999999.999~999999.999	-99999.9999~99999.9999	deg

Set the boundary coordinate values in the machine coordinate system of the positive and negative directions of the stored stroke detection 1 of each axis respectively. Set the outside of the boundary as a no-entry area. Only when the parameter 1300#2 (LMS) is "1" and the stored stroke limit switching signal EXLM (G7.6) is "1", the prohibited area set by this parameter is valid (the settings of parameter 1320 and parameter 1321) invalid).

Note 1: The axis of diameter programming must be set with the diameter value.

Note 2: When the parameter 1300#2 (LMS) is "0", or the stored travel limit switching signal EXLM (G7.6) is "0", this parameter is invalid. At this time, the prohibited area set in parameter 1320 and parameter 1321 is valid.

1.7 Parameters related to infeed speed

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000001

「Valid mode」: Immediate

#0 RPD Manual rapid operation during the period from power-on to reference point return

0: Invalid (change to manual feed)

1: Valid

#1 LRP Positioning (G00) is

0: Non-linear interpolation positioning

1: Linear interpolation positioning

#2 ZRL For machine coordinate positioning (G53)

0: Non-linear interpolation positioning

1: Linear interpolation positioning

#4 RF0 When cutting infeed speed override is 0% during rapid traverse

0: The tool does not stop moving

1: The tool stops moving

#5 TDR Dry Run during thread cutting or tapping

0: Valid

1: Invalid

#6 RDR Pair fast run command, dry run

0: Invalid

1: Valid

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 FVT Infeed override type

0: 4-bit

1: 8-bit

#2 JOV Manual override

0: Valid

1: Invalid (fixed at 100%)

#3 JVT JOG type override

0: 4-bit

1: 8-bit

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#5 HTG Spiral interpolation speed command

0: Specify with the tangent speed of the arc

1: Specify with the tangent speed including the linear axis

#7 RTV Thread cutting cycle tool run-out override

0: Valid

1: Invalid

	#7	#6	#5	#4	#3	#2	#1	#0
1404							DLF	

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#1 DLF After reference point is established, when manually returning to reference point

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

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

1410	Dry Running Speed (DRR)
-------------	--------------------------------

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 1000

「Valid mode」 : Immediate

「Input and output」 : Output related

「Value range」 :

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	mm/min	6~15000	6~12000	1000
Imperial machine	inch/min	6~6000	6~4800	

Set the speed during dry run.

1411

Feed Speed in Automatic Mode When Power Is Turned On (IFV)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Input/Output」: Input related

「Value range」:

Setting Unit	Unit	IS-B	IS-C
Metric input	mm/min	6~15000	6~12000
Imperial input	inch/min	6~6000	6~4800

Set the feed rate in automatic mode when the power is turned on. The unit is 1mm/min or 1inch/min or 1deg/min.

For machine tools that do not need to change the cutting speed during processing, this parameter can be used to specify the cutting feedrate, so that it is not necessary to specify the cutting feedrate in the program. But the actual feedrate is clamped by parameter 1422 (maximum cutting feedrate for all axes).

1420

Rapid Traverse Speed of Each Axis (RTT)

「Parameter type」: Word axis type-real number type

「Modification authority」: Machine tool

「Parameter default」: 8000

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1 mm/min	30~100000	6~60000	8000
Imperial machine	1 inch/min	30~48000	6~24000	
Rotary axis	1 deg/min	30~100000	6~60000	

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

1421

F0 Speed of Each Axis Rapid Traverse Override (F0R)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 400

「Valid mode」: Immediate

「Input and output」 : Output related

「Value range」 :

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1 mm/min	0~15000	0~12000	400
Imperial machine	1 inch/min	0~12000	0~6000	
Rotary axis	1 deg/min	30~15000	0~15000	

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

1422

Maximum Cutting Feed Rate of All Axes (MFR)

「Parameter type」 : Font type-real number type

「Modification authority」 : Machine tool

「Parameter default」 : 8000

「Valid mode」 : Immediate

「Input and output」 : Output related

「Value range」 :

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1mm/min	6~100000	6~60000	8000
Imperial machine	1inch/min	6~48000	6~24000	

Set the maximum cutting feedrate for all axes.

1423

Manual Feed Rate of Each Axis (JFR)

「Parameter type」 : Word axis type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 1000

「Valid mode」 : Immediate

「Input and output」 : Output related

「Value range」 :

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1mm/min	6~100000	6~60000	1000
Imperial machine	1inch/min	6~48000	6~24000	
Rotary axis	1dev/min	6~100000	6~60000	

Set the feed rate during manual continuous feed (manual feed) of each axis. The actual feed rate is clamped by parameter No.1422 (maximum cutting feed rate for all axes).

1424

Manual Rapid Traverse Speed of Each Axis (MRR)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 8000

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1 mm/min	30~100000	30~60000	8000
Imperial machine	1 inch/min	30~48000	30~24000	
Rotary axis	1 deg/min	30~100000	30~60000	

When the rapid traverse override is set to 100%, the manual rapid traverse speed of each axis is equal to the maximum MPG feed speed.

Note: If set to 0, use the set value of parameter 1420.

1425

FL Speed of Reference Point Return of Each Axis (FLR)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 200

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1 mm/min	6~15000	6~12000	200
Imperial machine	1 inch/min	6~12000	6~6000	
Rotary axis	1 deg/min	6~15000	6~12000	

Set the speed (FL speed) of each axis after deceleration when returning to the reference point.

1428

Reference Point Return Speed of Each Axis (RPF)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 5000

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1 mm/min	0, 6~100000	0, 6~60000	5000
Imperial machine	1 inch/min	0, 6~48000	0, 6~24000	
Rotary axis	1 deg/min	0, 6~100000	0, 6~60000	

Set the rapid traverse speed for the case of reference point return using a deceleration block, or for the case of reference point return when the reference point has not yet been established. When the parameter value is 0, parameter №1421 is valid.

1434

The Maximum Feed Rate of Manual MPG Feed for Each Axis (HMF)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 5000

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1 mm/min	0, 6~100000	0, 6~60000	5000
Imperial machine	1 inch/min	0, 6~48000	0, 6~24000	
Rotary axis	1 deg/min	0, 6~100000	0, 6~60000	

Set the maximum feed rate of manual MPG feed for each axis. When it is set to 0, the setting value of parameter No. 1424 is valid.

1466

Infeed Speed (FRT) When Performing Thread Cutting Retracting Motion

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 8000

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1 mm/min	0, 6~100000	0, 6~60000	8000
Imperial machine	1 inch/min	0, 6~48000	0, 6~24000	
Rotary axis	1 deg/min	0, 6~100000	0, 6~60000	

Set the feed rate of the retracting action during thread cutting. When the parameter setting value is "0", the retracting action will be performed at the speed of the long axis.

1467

Speed Override (FRV) When Executing Thread Cutting Run-out Motion

「Parameter type」: Word axis type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」:

Set the speed override value during thread cutting run-out action. It is valid when the parameter THV (No.1610.7) is set to "1". When the parameter setting value is "0", the run-out work will be performed at 1 times the thread speed, Unit: 0.1 times.

1468

Error Compensation Value During Thread Machining(ICD)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~50000

Set the error compensation values (0~50000) for different rotational speeds during thread machining.

1469

Check Spindle Speed Fluctuation When Starting Thread Cutting(TSP)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value Ranges」 : (1-100)%

Set the spindle speed fluctuation percentage when thread cutting starts. When the set value is 0, it will not be detected.

1480

Set the Error Range of Machining Track in G64 Mode (ARE)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Input/Output」 : Input related

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~9.999	0~9.9999	mm
Imperial input	0~0.9999	0~0.99999	inch

This parameter sets the machining track error range in G64 mode. When it is set to 0, it means that the track error is not detected (0~9.9999).

1.8 Parameters related to acceleration and deceleration control

1601

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

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 TSS Thread machining mode selection

0: Mode A

1: Mode B

#4 RTO During rapid operation, the block

0: No overlap

1: Overlap

	#7	#6	#5	#4	#3	#2	#1	#0
1610	THVx	THTx	THLx	JGLx		RTSx		

「Parameter type」: Bit axis type

「Modify authority」: Equipment

「Parameter default」: 01110100

「Valid mode」: Immediate

#2 RTSx Acceleration and deceleration type of rapid feed

0: Linear acceleration and deceleration

1: S-type acceleration/deceleration

#4 JGLx Acceleration/Deceleration of manual feed is

0: Exponential acceleration/deceleration

1: Linear acceleration/deceleration after interpolation

#5 THLx Acceleration/Deceleration of retreat during thread cutting is adopted

0: Exponential acceleration/deceleration

1: Linear acceleration/deceleration

#6 THTx Acceleration/Deceleration of retreat during thread cutting is adopted

0: 1610#5 parameter is valid

1: S-type acceleration/deceleration

#7 THVx Speed override for the retraction action during thread cutting

0: Invalid

1: Valid

1620

Linear Acceleration/Deceleration Time Constant T of Fast Forward on Each Axis(TT1)

「Parameter type」: Word axis type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 0~4000

Set the acceleration/deceleration time constant for rapid feed.

1622

Acceleration/Deceleration Time Constant of Cutting Feed after Interpolation of Each Axis (ATC)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~4000

Set the exponential or linear acceleration/deceleration time constant after interpolation for each axis cutting feed.

The specific use type is selected by parameter 1610#0 (CTLx). If the CTLx setting is linear acceleration/deceleration type after direct interpolation, the maximum time constant of acceleration/deceleration will be limited within 512ms, and all values exceeding the limit are processed with a time of 512ms.

Note: Except for special purposes of this parameter, all axes must be set to the same time constant. If different time constants are set, it will be impossible to get the correct straight line or arc shape.

1624

Acceleration/Deceleration Time Constant of Each Axis Manual Feed after Interpolation (JET)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~4000

Set the manual feed exponential type or linear acceleration/deceleration time constant after interpolation for each axis.

The specific use type is selected by parameter 1610#4 (JGLx). If JGLx is set to linear acceleration/deceleration type after direct interpolation, the maximum time constant of acceleration/deceleration will be limited within 512ms, and all values exceeding the limit are processed with a time of 512ms.

1625

FL Speed of Exponential Acceleration/Deceleration for Manual Feed of Each Axis(FLJ)

「Parameter type」 : Word axis type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 30

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1 mm/min	0, 6~15000	0, 6~12000	30
Imperial machine	1 inch/min	0, 6~12000	0, 6~6000	
Rotary axis	1 deg/min	0, 6~15000	0, 6~12000	

Set the lower limit speed (FL speed) of exponential acceleration/deceleration for manual feed of each axis.

1626

Acceleration/Deceleration Time Constant During Thread Cutting Cycle of Each Axis(TET)

「Parameter type」: Word axis type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 0~4000

Set the linear and exponential acceleration/deceleration time constants for each axis thread cutting cycle.

1627

FL Speed of Exponential Acceleration/Deceleration of Thread Cutting Cycle of Each Axis (FLT)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 30

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1 mm/min	0, 6~15000	0, 6~12000	30
Imperial machine	1 inch/min	0, 6~12000	0, 6~6000	
Rotary axis	1 deg/min	0, 6~15000	0, 6~12000	

Set the lower limit speed (FL speed) of exponential acceleration/deceleration of thread cutting cycle of each axis .

1628

Acceleration/Deceleration Time Constant of Retract during Thread Cutting Cycle of Each Axis(TST)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~4000

Set the acceleration/deceleration time constant for the minor axis of the retraction action during each axis's thread cutting cycle. When the setting value of this parameter is "0", the value of parameter No. 1626 (0~4000ms) is used.

1630

Acceleration/Deceleration Time Constant of MPG Feed of Each Axis (MET)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~4000

Set acceleration/deceleration time constant (0~4000) ms of MPG feed of each axis.

1.9 Parameters Related to servo and backlash compensation

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

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 10000000

「Valid mode」 : Immediate

#6 BD8 Pulse output frequency of backlash compensation

0: Compensate with the frequency set by parameter #1853

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

#7 BDEC Backlash compensation method

0: Output at a fixed pulse frequency (set by parameters #1853 and #1800.6)

1: Pulse frequency is output according to acceleration and deceleration feature

	#7	#6	#5	#4	#3	#2	#1	#0
1811	NANOx					POD	ABPx	ISAx

「Parameter type」: Bit axis type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#0 ISAx Whether servo alarm signal input valid

0: Valid

1: Invalid

#1 ABPx Pulse drive mode selection

0: Pulse + direction mode

1: AB phase pulse mode

#2 POD Output direction selection of each axis pulse

0: No negation

1: Negate

#7 NANOx Whether nanometer interpolation is valid for each axis

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
1812								RCAx

「Parameter type」: Bit axis type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#0 RCAx Removal function for each control axis

0: Invalid

1: Valid

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

「Parameter type」: Bit axis type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#0 APTx Whether the absolute position detector uses a linear scale

0: Not used

1: Used

#4 APZx Position of mechanical position and absolute position detector when the absolute position detector is used

0: Inconsistent

1: Consistent

Note: When using an absolute position detector, this parameter must be set to 0 when performing initial adjustment or after replacing the absolute position detector. Turn off the power and then turn on the power to return to the reference point manually. In this way, the mechanical position is consistent with that of the position detector, and this parameter will be automatically set to 1.

#5 APCx Position Detector

0: Do not use absolute position detector

1: Use absolute position detector (absolute pulse encoder)

1816

Detection Multiplication Ratio of Each Axis (DMR)

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」 : 1~99999999

Set the detection multiplication ratio (DMR) of each axis.

1820

Command Multiplication Ratio of Each Axis (CMR)

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」 : 1~99999999

Gear ratio of each axis output = CMR/ DMR

Detection unit = minimum movement unit / CMR

The relationship between the setting unit and the minimum movement unit is as follows.

		IS—B		IS—C	
	Input	Minimum Setting Unit	Minimum Movement Unit	Minimum Setting Unit	Minimum Movement Unit
Metric machine	Metric	0.001mm (diameter)	0.0005mm	0.0001 mm (diameter)	0.00005mm
		0.001mm (radius)	0.001mm	0.0001 mm (radius)	0.0001mm
	Imperial	0.0001 inch (diameter)	0.0005mm	0.00001 inch (diameter)	0.00005mm
		0.0001 inch (radius)	0.001mm	0.00001 inch (radius)	0.0001mm
Imperial machine	Metric	0.001mm (diameter)	0.00005 inch	0.0001 mm (diameter)	0.000005 inch
		0.001mm (radius)	0.0001 inch	0.0001 mm (radius)	0.00001 inch
	Imperial	0.0001 inch (diameter)	0.00005 inch	0.00001 inch (diameter)	0.000005 inch
		0.0001 inch (radius)	0.0001 inch	0.00001 inch (radius)	0.00001 inch
Rotary axis		0.001 deg	0.001deg	0.0001 deg	0.0001deg

1826

In-position Width During Rapid Feed of Each Axis (PWR)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory
		IS-B	IS-C	Default
Metric machine	mm	0~99999.999	0~9999.9999	0
Imperial machine	inch	0~9999.9999	0~999.99999	
Rotary axis	deg	0~99999.999	0~9999.9999	

Set the in-position width of each axis during rapid feed (0~9999.9999). When the set value is 0, the in-position detection will not be performed.

1827

In-position Width During Cutting Feed of Each Axis (PWF)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	mm	0~99999.999	0~9999.9999	0
Imperial machine	inch	0~9999.9999	0~999.99999	
Rotary axis	deg	0~99999.999	0~9999.9999	

Set the in-position width (0~99999999) during the cutting feed of each axis. When the set value is 0, the in-position detection will not be performed.

1851

Backlash Compensation Amount of Each Axis (BCV)

「Parameter type」 : Word axis type-real number type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	mm	-1.000~1.000	-1.0000~1.0000	0
Imperial machine	inch	-1.0000~1.0000	-1.00000~1.00000	
Rotary axis	deg	-1.000~1.000	-1.0000~1.0000	

Set the amount of backlash compensation of each axis.

After power-on, when the machine tool moves in the opposite direction of reference point return, the first backlash compensation is performed.

The detection unit is related to the parameter 1820 (command multiplication ratio CMR) and the minimum movement unit. For the relationship between the setting unit and the minimum movement unit, see the note of parameter 1820.

1853

Setting Value of Backlash Compensation Pulse Frequency

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 16

「Valid mode」 : Immediate

「Value range」 : 1~128

Setting value of backlash compensation pulse frequency (1~128)

1859

Effective Time Constant of Backlash Acceleration of Each Axis (BAT)

「Parameter type」: Word axis type

「Modification authority」: Machine tool

「Parameter default」: 40

「Valid mode」: Immediate

「Value range」: 0~200 ms

Set the effective time constant of backlash acceleration/deceleration of each axis.

1.10 parameters related to input and output

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Immediate

- #0 ITL Makes all axes interlock signal**
 0: Invalid
 1: Valid
- #2 ITX Makes each axis interlock signal**
 0: Invalid
 1: Valid
- #7 ESP External emergency stop alarm input signal (X0.5)**
 0: Emergency stop alarms when the signal is 0 (low level)
 1: Emergency stop alarms when the signal is 1 (high level)

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00100000

「Valid mode」: Immediate

- #0 BSL Make the block and cutting block start interlock signal**
 0: Invalid
 1: Valid
- #5 OTH Overtravel limit signal**
 0: Check

1: Do not check

Note: After the overtravel alarm is generated, if you modify this parameter to 1 (no check), and then press reset, the alarm cannot be cleared. You need to manually move to the stroke, and then set this parameter to 0 to clear the alarm.

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 GDC Deceleration signal of reference point return

0: Use X signal

1: Use G196 (X signal is invalid)

#1 EPN In the external part number search, select the signal used to specify the part number

0: Use signal PN1~PN16

1: Use extended signals EPN0~EPN13

#2 EPS Start signal for external part number search

0: Use automatic operation start signal ST

1: Use the external part number to retrieve the start signal EPNS

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Immediate

#1 ASG Allocate addresses for skip signals, multi-step skip signals, and measurement position arrival signals. Valid only when bit #2 of parameter 3008 is set to 1. The specific address is set by parameter 3012.

0: Signal address X

1: Signal address G

#2 XSG Allocate addresses for skip signals, multi-step skip signals, and measurement position arrival signals.

- 0: Belongs to a fixed address X0
- 1: Determined by parameter 3008#1

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

- 「Parameter type」 : Bit axis type
- 「Modification authority」 : Machine tool
- 「Parameter default」 : 00100000
- 「Valid mode」 : Immediate

#5 DECx Return to reference point deceleration signal

- 0: Decelerate when the signal is 0 (low level)
- 1: Decelerate when the signal is 1 (high level)

3010	Delay Time of Strobe Signal MF, TF, SF (MFT)
------	--

- 「Parameter type」 : Font type
- 「Modification authority」 : Machine tool
- 「Parameter default」 : 16
- 「Valid mode」 : Immediate
- 「Value range」 : 16ms~32767ms

Set the time from the sending of M, S, T, and B codes to the sending of MF, SF, TF, and BF.

3011	Minimum Width (MAW) of Completion Signal (FIN) of M, T, S
------	---

- 「Parameter type」 : Font type
- 「Modification authority」 : Machine tool
- 「Parameter default」 : 24
- 「Valid mode」 : Immediate
- 「Value range」 : 16ms~32767 ms

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

Note: The time is set by 8ms. If the set value is not a multiple of 8, the carry is a multiple of 8.

3012	Distribute Skip Signal, Multi-Step Skip Signal and The Address of The Measurement Position Arrival Signal
------	---

- 「Parameter type」 : Font type
- 「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~511

Set the X address (0~127) or G address (0~511) used to allocate skip signals, multi-step skip signals, and measurement position arrival signals.

3013

**Distribute X address of Deceleration Signal Used for Reference Point
Return Operation**

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~127

Set the X address (0~127) of the deceleration signal used for the reference point return operation.

3014

**Distribute Bit Address of X Address of Deceleration Signal Used for
Reference Point Return Operation**

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~7

Set the bit address (0~7) of the X address of deceleration signal used for reference point return operation.

3017

Reset Signal Output Time (RST)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 32

「Valid mode」 : Immediate

「Value range」 : 0~400 ms

Set the delay time when the reset signal RST is output.

RST signal output time = reset time + this parameter value × 16ms.

3019

Assign Address of Tool Compensation Amount write Signal (XSG2)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~127

Set the address of the tool compensation amount write signal used to allocate the X address.

3020

Assign Bit Address of Skip Signal SKIP

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~7

Set the bit address used to allocate the skip signal SKIP.

3021

Assign Bit Address of Multi-step Skip Signal SKIP2

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~7

Set the bit address used to allocate the multi-step skip signal SKIP2.

3022

Assign Bit Address of Multi-step Skip Signal SKIP3

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~7

Set the bit address used to allocate the multi-step skip signal SKIP3.

3023

Assign Bit Address of Multi-step Skip Signal SKIP4

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~7

Set the bit address used to allocate the multi-step skip signal SKIP4.

3025

Allocate Addresses For Multi-Step Skip Signals (P5~P12) (SKPEX)

「Parameter type」 : Font type

「Modification authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~127

Set the X address (0~127) used to allocate multi-step skip signals (P5~P12).

3030

Allowable Digits of M Code (MCB)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 2

「Valid mode」 : Immediate

「Value range」 : 2~4

Set the allowable digits of the M code.

3031

Allowable Digits of S Code (SCB)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 4

「Valid mode」 : Immediate

「Value range」 : 1~5

Set the allowable digits of the S code (up to 5 digits are allowed).

3032

Allowable Digits of T Code (TCB)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 4

「Valid mode」 : Immediate

「Value range」 : 2~8

Set the allowable digits of the T code.

3033

Allowable Digits of B Code (BCN)

「Parameter type」 : Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~8

Allowable digits (0~8) of B code (second auxiliary function).

3050

The Number of I/O Units Controlled by the System (IOMAX)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 1

「Valid mode」: Power on

「Value range」: 0~4

Set the number of I/O units controlled by the system (up to 4).

3051

Logical ID Number of System Control I/O Unit 1 (IOID1)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 100

「Valid mode」: Power on

「Value range」: 0,100~110

Set the logical ID number of the system control I/O unit 1 (0 means that the I/O unit is not connected to GSK-Link).

3052

Logical ID Number of System Control I/O Unit 2 (IOID2)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 100

「Valid mode」: Power on

「Value range」: 0,100~110

Set the logical ID number of the system control I/O unit 2 (0 means that the I/O unit is not connected to GSK-Link).

3053

The Logical ID Number of The System Control I/O Unit 3 (IOID3)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 100

「Valid mode」 : Power on

「Value range」 : 0,100~110

Set the logical ID number of the system control I/O unit 3 (0 means that the I/O unit is not connected to GSK-Link).

3054

Logical ID Number of System Control I/O Unit 4 (I/OID4)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 100

「Valid mode」 : Power on

「Value range」 : 0,100~110

Set the logical ID number of the system control I/O unit 4 (0 means that the I/O unit is not connected to GSK-Link).

3060

Number of Gateways Controlled by System (GWN)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~2

This parameter sets the number of gateways controlled by the system (0 means no gateway is used).

3061

Station Address of System Control Gateway 1 (GWA1)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0,200~254

This parameter sets the station address of the system control gateway 1 (0 means that the gateway is not used).

3062

Station Address of System Control Gateway 2 (GWA2)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」: 0,200~254

This parameter sets the station address of the system control gateway 2 (0 means no gateway is used).

	#7	#6	#5	#4	#3	#2	#1	#0
3071								GWC1

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#0 GWC1 Whether gateway 1 data uses CRC check or not

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
3072								GWC2

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#0 GWC2 Whether gateway 2 data uses CRC check or not

0: Invalid

1: Valid

1.11 Parameters related to display and editing

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 EHA Run program when the edited program is not loaded and the program cursor is not at the first line

0: Prompt the user

1: Do not prompt

#4 BGD When background editor selects the program that has been selected in the foreground

0: Editable

1: Not editable

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 11000000

「Valid mode」: Immediate

#0 MCN Machine position display

0: Display according to the output unit

(It has nothing to do with whether the input is metric or imperial. Metric machine tools are displayed in metric units, and Imperial machine tools are displayed in imperial units)

1: Display according to the input unit

(For metric input, display in metric system; when input in inch system, display in English system)

#4 DRL Display of relative position

0: Display the actual position with tool offset

1: Display the programmed position without tool offset

Note: When using the moving coordinate system for tool shape compensation (parameter 5002#4 (LGT) is 0), the displayed position is the programmed position that ignores the tool compensation (set this parameter to 1). However, the programmed position that does not include the tool contour compensation amount cannot be displayed.

#5 DRC Display of relative position

0: Display the actual position with tool nose radius compensation

1: Display the programmed position without tool nose radius compensation

#6 DAL Absolute position display

0: Display the actual position with tool offset

1: Display the programmed position without tool offset

Note: When using the moving coordinate system for tool shape compensation (parameter 5002#4 (LGT) is 0), the displayed position is the programmed position ignoring the tool compensation (this parameter is set to 1), but it cannot be displayed without the tool shape compensation amount. Programming location.

#7 DAC Absolute position display

0: Display the actual position with tool nose radius compensation

1: Display the programmed position without tool nose radius compensation

	#7	#6	#5	#4	#3	#2	#1	#0
3107					REV	DNC		MIF

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00010000

「Valid mode」: Immediate

#0 MIF Minimum value displayed by the f command is

0: 1mm/min or 0.01inch/min

1: 0.001mm/min or 0.00001inch/min

#2 DNC Reset whether to clear the display of DNC running program

0: Do not clear

1: Clear

#3 REV Displays actual speed in infeed mode per revolution

0: mm/min or inch/min

1: mm/rev or inch/rev

	#7	#6	#5	#4	#3	#2	#1	#0
3110	OPH			TSPIT		AHC		

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000100

「Valid mode」: Power on

#2 AHC Can alarm history be cleared by soft key

0: Yes

1: No

#4 TSPIT Tool offset and wear data page

0: Combined display

1: Separate display

#7 OPH Operation log and processing log page

0: Be displayed

1: Do not display

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

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#4 SPA Whether to automatically switch to the custom screen at startup

0: Do not switch

1: Switch

#7 NPA Whether switch to alarm/information screen when alarm occurs and when operating information is input

0: Do not switch

1: Switch

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

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 IPC Under current page, when press current page function key

0: Switch screen

1: Do not switch screen

	#7	#6	#5	#4	#3	#2	#1	#0
3115	ASTx					SRTx		NDPx

「Parameter type」 : Bit axis type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 NDPx Whether to display current position

0: Display

1: Do not display

#2 SRTx Axis coordinate display

0: Normal display

1: Auto-tuning to integer

#7 AST x Axis movement status display

0: Do not display

1: Display the current motion state of the axis

3123

Screen Saver Activation Time (CLRT)

「Parameter type」: Font type

「Modification authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~120

Set the screen saver start time (0~120 minutes)

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

3202

CPD

NE8

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 NE8 Whether to prohibit editing, deleting, modifying, copying and other operations of program numbers 8000~8999

0: Not prohibited

1: Prohibited

#5 CPD When deleting NC program, confirm the message and confirm soft key

0: Do not display

1: Display

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

3203

MCL

MER

RCPL

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 RCPL In the non-editing mode, whether the reset operation will return the cursor to the first line of the program

0: Return

1: Do not return

#6 MER When running in single block mode under MDI (Manual Data Input), after the last block in the program has been executed, the executed program

0: Do not delete

1: Delete

Note: Even if the setting is not to be deleted (MER is 0), when "%" (end code)" is read in and executed, the program is also deleted ("% is automatically inserted at the end of the program).

#7 MCL Whether a program edited in MDI mode is deleted by a reset operation.

0: Do not delete

1: Delete

	#7	#6	#5	#4	#3	#2	#1	#0
3209	PRO	PSS						MPD

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

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

0: Do not display

1: Display

#6 PSS Whether the PLC window command is allowed to write parameters when parameters are locked

0: Not allowed

1: Allowed

#7 PRO Parameter protection lock and program protection lock

0: Share G46.3

1: Use separately

3216	Increment Value Of Sequence Numbers When Automatically Inserting Sequence Numbers (INC)
-------------	--

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」 : 10

「Valid mode」 : Immediate

「Value range」 : 1~9999

When the sequence number is automatically inserted (parameter 0000#5 (SEQ) is 1), the incremental value of the sequence number of each block.

3281

Setting Interface Display Language (LANG)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

0 : English 1 : Chinese 2 : Russian 3 : French 4:Spain 5:Portugal 6:Polish

3282

The Number of Days to Be Notified In Advance of The Time-Limited Shutdown (NDAYS)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 3

「Valid mode」 : Immediate

「Value range」 : 1~30

1.12 Parameters related to programming

	#7	#6	#5	#4	#3	#2	#1	#0
3401		GSB					CPQ	

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Power on

#1 CPQ Program command PQ value unit

0: mm or degree

1: Minimum command unit

#6 GSB Set the format of G code

0: G code system A

1: G code system B

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR		FPM	G91			G01

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 01010000

「Valid mode」 : Immediate

#0 G01 Modal when the power is turned on

0: G00 mode (positioning)

1: G01 mode (linear interpolation)

#3 G91 In G code system B, the system defaults to

0: G90 mode (absolute command)

1: G91 mode (incremental command)

#4 FPM The system defaults to

0: Feed per revolution

1: Feed per minute

#6 CLR Press the reset button, external reset signal and emergency stop, G code modal and feed speed

0: Maintain modal

1: Switch to power-on state

#7 G23 When the power is turned on, it is

0: G22 mode (check stored stroke)

1: G23 mode (do not perform storage travel check)

	#7	#6	#5	#4	#3	#2	#1	#0
3403		AD2	CIR	RER				

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Power on

#4 RER In circular interpolation, when R is too small, the end point is not on the arc, and the radius is not out of tolerance

0: Calculate the new radius, and the trajectory is a semicircle

1: P/S alarm occurs

#5 CIR In circular interpolation (G02, G03) commands, when neither the distance from the starting point to the center (I, J, K) nor the arc radius is specified

0: Linear interpolation moves to the end point

1: P/S alarm occurs

#6 AD2 In the same block, when 2 or more of the same addresses are commanded

0: The following command is valid

1: It is regarded as a program error, and a P/S alarm occurs

Note: When this parameter is 1, an alarm occurs when two or more G codes of the same group are commanded in the same block.

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B	EOR	M02	M30	M99			

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#3 M99 Processing of M99 command in automatic operation

0: Not sent to PLC

1: Sent to PLC

#4 M30 In automatic operation, the processing of M30 commands

0: Return to the beginning of the program

1: Do not return to the beginning of the program.

#5 M02 In automatic operation, the processing of M02 command

0: Return to the beginning of the program

1: Do not return to the beginning of the program.

#6 EOR When reading "%" (end of program) during program execution

0: Alarm

1: No alarm

Note: When executing "%" (end of program), CNC resets, but does not turn off auxiliary function output.

#7 M3B The number of M codes that can be commanded in the same Block

0: 1

1: Up to 3

	#7	#6	#5	#4	#3	#2	#1	#0
3405			DDP	CDR				

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#4 CDR The address used in the commands for direct input of chamfering and drawing size

0: Use with commas on "C", "R", and "A"

1: Use without comma

#5 DDP Angle command for direct input of drawing size

0: Normal specifications

1: Command supplementary angle

The functions with comma and without comma are the same. Note that: when the parameter is set to use the chamfering function without comma, when R is in the same segment with the arc command G02/G03, R is the radius value and the chamfering function is invalid .

	#7	#6	#5	#4	#3	#2	#1	#0
3408								AXF

「Parameter type」: Bit axis type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 AXF Whether the hidden axis set by parameter 3115#0 allows programming

0: Allow

1: Not allowed

3410	Permissible Error of Arc Radius (CRE)
-------------	--

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS—B	IS—C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the allowable error value of the starting point radius and the ending point radius of circular interpolation (G02, G03). When the radius difference of circular interpolation is greater than the set value, a P/S alarm will appear.

Note: When the set value is 0, the arc radius difference check is not performed.

3411	M Code 1 to Prevent Buffering (BLKM1)
3412	M Code 2 to Prevent Buffering (BLKM2)
3413	M Code 3 to Prevent Buffering (BLKM3)
3414	M Code 4 to Prevent Buffering (BLKM4)
3415	M Code 5 to Prevent Buffering (BLKM5)
3416	M Code 6 to Prevent Buffering (BLKM6)
3417	M Code 7 to Prevent Buffering (BLKM7)
3418	M Code 8 to Prevent Buffering (BLKM8)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~9999

This parameter sets the M code that prevents buffering. Before ending the processing of the M function on the machine side, if the machine must execute the operation processing specified by the M code, set this code.

3460	Command Address of the Second Auxiliary Function (BCA)
------	---

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0, 65~67, 85~87

The command address of the second auxiliary function (0, 65~67, 85~87) 0 is to close the second auxiliary function.

3470

The Reference Value of The Programming Movement of Each Spindle in The Position Mode (CSM)

「Parameter type」: Word spindle type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS—B	IS—C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the reference value (0~9999.9999) of the programming movement amount of each spindle in the position mode.

1.13 Parallel program

3500

#7	#6	#5	#4	#3	#2	#1	#0
				PPT4	PPT3	PPT2	PPT1

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 PPT1 The first parallel program content

0: Determined by parameter P3510

1: Determined by the main program

#1 PPT2 The 2nd parallel program content

0: Determined by parameter P3511

1: Determined by the main program

#2 PPT3 The 3rd parallel program content

0: Determined by parameter P3512

1: Determined by the main program

#3 PPT4 The 4th parallel program content

0: Determined by parameter P3513

1: Determined by the main program

3501

#7	#6	#5	#4	#3	#2	#1	#0
				PG844	PG843	PG842	PG841

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 PG841 The first parallel program command rigid tapping method

0: M code for commanding rigid tapping

1: G84/G88 as the G code for rigid tapping

#1 PG842 The second parallel program command rigid tapping method

0: M code for commanding rigid tapping

1: G84/G88 as the G code for rigid tapping

#2 PG843 The third parallel program command rigid tapping method

0: M code for commanding rigid tapping

1: G84/G88 as the G code for rigid tapping

#3 PG844 The fourth parallel program command rigid tapping method

0: M code for commanding rigid tapping

1: G84/G88 as the G code for rigid tapping

	#7	#6	#5	#4	#3	#2	#1	#0
3502					PCP4	PCP3	PCP2	PCP1

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 PCP1 When the address Q is commanded during rigid tapping and drilling in the first parallel program

0: High-speed deep hole tapping cycle

1: Deep hole tapping cycle

#1 PCP2 When the address Q is commanded during rigid tapping and drilling in the second parallel program

0: High-speed deep hole tapping cycle

1: Deep hole tapping cycle

#2 PCP3 When the address Q is commanded during rigid tapping and drilling in the third parallel program

0: High-speed deep hole tapping cycle

1: Deep hole tapping cycle

#3 PCP4 When the address Q is commanded during rigid tapping and drilling in the fourth parallel program

0: High-speed deep hole tapping cycle

1: Deep hole tapping cycle

	#7	#6	#5	#4	#3	#2	#1	#0
3508								SEN

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Power on

#0 SEN Whether to display the parallel program page

0: Do not display

1: Display

3510	The first parallel program name (PSEN1)
------	---

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~9999

This parameter sets the name of the parallel program executed by signal G280.0, 0 means the function is invalid.

3511	The Second Parallel Program Name (PSEN2)
------	--

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~9999

This parameter sets the name of the parallel program executed by the signal G280.1, 0 means the function is invalid.

3512	The Third Parallel Program Name (PSEN3)
------	---

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~9999

This parameter sets the name of the parallel program executed by the signal G280.2, 0 means the function is invalid.

3513

The Fourth Parallel Program Name (PSEN4)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~9999

This parameter sets the name of the parallel program executed by the signal G280.3, 0 means the function is invalid.

3515

The Serial Number Of The Parallel Program Where Each Axis Is Located (PSNA)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~4

Set the serial number of the parallel program where each axis is located. Setting it to 0 indicates invalidity.

3516

The Programming Name Of Each Axis In The Parallel Program (PCAN)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 88,89,90,65,66,67,85,87

Set the axis names for each axis in the parallel program (X=88, Y=89, Z=90, A=65, B=66, C=67, U=85, V=86, W=87).

3520

Minimum Value Of The Range For Parallel Program Waiting M Codes (PWMI)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0,100~999

Set the minimum value of the range for waiting M codes between parallel programs and the main program (0, 100~999).

3521

**Maximum Value Of The Range For Parallel Program Waiting M Codes
(PWMM)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0,100~999

Set the maximum value of the range for waiting M codes between parallel programs and the main program (0, 100~999).

3530

**M Code for Rigid Tapping of The First Parallel Program Command
(RTMC1)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~255

Set the M code (0~255) that specifies the rigid tapping method. When it is set to 0, the system defaults to M29.

3531

**M Code for Rigid Tapping in The Second Parallel Program Command
(RTMC2)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~255

Set the M code (0~255) that specifies the rigid tapping method. When it is set to 0, the system defaults to M29.

3532

**M Code for Rigid Tapping of The Third Parallel Program Command
(RTMC3)**

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~255

Set the M code (0~255) that specifies the rigid tapping method. When it is set to 0, the system defaults to M29.

3533

**M Code for Rigid Tapping of The Fourth Parallel Program Command
(RTMC4)**

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~255

Set the M code (0~255) that specifies the rigid tapping method. When it is set to 0, the system defaults to M29.

3540

**The First Parallel Program Deep Hole Tapping And Drilling Return
Amount (PRTRD1)**

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the retraction amount of the high-speed deep hole cycle, or the empty stroke amount of the deep hole cycle (0~9999.9999).

3541

**The Second Parallel Program Deep Hole Tapping And Drilling Return
Amount (PRTRD2)**

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input/Output」 : Input related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the retraction amount of the high-speed deep hole cycle, or the empty stroke amount of the deep hole cycle (0~9999.9999).

3542

The Third Parallel Program Deep Hole Tapping And Drilling Return Amount (PRTRD3)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input/Output」 : Input related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the retraction amount of the high-speed deep hole cycle, or the empty stroke amount of the deep hole cycle (0~9999.9999).

3543

The Fourth parallel program deep hole tapping and drilling return amount (PRTRD4)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input/Output」 : Input related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the retraction amount of the high-speed deep hole cycle, or the empty stroke amount of the deep hole cycle (0~9999.9999).

1.14 Parameters related to pitch error compensation

	#7	#6	#5	#4	#3	#2	#1	#0
3600					PVH			NPE

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#0 NPE Whether to Use Pitch Error Compensation

0: Used

1: Not used

#3 PVH Pitch error compensation display

0: Horizontal

1: Vertical

	#7	#6	#5	#4	#3	#2	#1	#0
3605							IPPx	BDPx

「Parameter type」: Bit axis type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#0 BDPx Whether to Use Two-Way Pitch Error Compensation

0: Not used

1: Used

#1 IPPx Whether to Use Interpolation Type Pitch Error Compensation

0: Not used

1: Used

3620	Pitch Error Compensation Number Of Each Axis Reference Point(NPR)
-------------	--

「Parameter type」: Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~199

3621

**The Number Of The Pitch Error Compensation Point At The Farthest
End In The Negative Direction Of Each Axis (NEN)**

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~199

This parameter sets the number of the farthest pitch error compensation point in the negative direction of each axis.

3622

**The Number Of The Pitch Error Compensation Point At The Farthest
End In The Positive Direction Of Each Axis (NEP)**

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~199

This parameter sets the number of the farthest pitch error compensation point in the positive direction of each axis.

Note: The setting value of this parameter is larger than the setting value of parameter NO.3620.

3623

Pitch Error Compensation Magnification Of Each Axis (PCM)

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」 : 0~100

Set the magnification of each axis's pitch error compensation (0~100), that is, compensation value * compensation magnification.

3624

Pitch Error Compensation Point Spacing Of Each Axis (PCI)

「Parameter type」 : Word axis type-real number type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Input and output」 : Output related

「Valid mode」 : Power on

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	0~99999.999	0~9999.9999	mm
Imperial machine	0~9999.9999	0~999.99999	inch
Rotary axis	0~99999.999	0~9999.9999	deg

The pitch compensation points are distributed at equal intervals, and the interval value is set separately for each axis. The minimum value of the distance is limited and is determined by the following formula:
minimum value = maximum feedrate (rapid feedrate)/7500.

Unit: Minimum pitch of screw compensation: mm, inch, deg.

Maximum feed rate: mm/min, inch/min, deg/min.

[Example] The minimum value of the pitch error compensation interval when the maximum feed speed is 15000mm/min is 2mm.

However, according to the set magnification, when the absolute value of the compensation amount of the compensation point exceeds 100, the magnification calculated by the following formula will enlarge the interval of the compensation point.

Multiple = maximum compensation amount (absolute value) / 128 (numbers after the decimal point are rounded)

The minimum pitch of pitch compensation = the value obtained from the above-mentioned maximum feedrate × multiple.

Note: The unit of the pitch compensation value is the same as the detection unit. The detection unit is related to parameter 1820 (command multiplier ratio CMR) and the minimum movement unit. The relationship between the setting unit and the minimum movement unit can be found in the annotation of parameter 1820.

3627

The Pitch Error Compensation Value Of The Reference Point When Moving From The Direction Opposite To The Return-To-Origin Direction To The Reference Point (PCD)

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : -99999~99999

When setting the origin direction as positive/negative direction, set the pitch error compensation amount in the reference point when moving from the negative/positive direction as an absolute value.

3628

Setting Value Of Pitch Compensation Pulse Frequency (NPF)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 8

「Valid mode」 : Immediate

「Value range」 : 1~64

The setting value of pitch compensation pulse frequency (1~64).

1.15 Parameters related to spindle control

	#7	#6	#5	#4	#3	#2	#1	#0
3700						CSB		CSC

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 CSC Whether to Clear the Coordinate Value when the CS Contour Control is Switched to the Spindle Mode (the parameter is valid when the 3700#2 Parameter is Set to 0)

0: Do not clear

1: Clear

#2 CSB Whether to Automatically Establish a Coordinate System when Switching the CS Contour Control to the Position Mode

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
3701	SCSs							SSDs

「Parameter type」 : Bit spindle type

「Modification authority」 : Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#0 SSDs Spindle speed display

0: Actual speed

1: Command speed

#7 SCSs Whether the Spindle Cs Contour Control Function is Valid

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
3702								RSF

「Parameter type」: Bit spindle type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#0 RSF Spindle removal function for each spindle

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
3703			MPR		MPP			

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Power on

#3 MPP In the Multi-spindle Control, whether to Use the Program Command to Replace the Signal SWS to Select the Spindle

0: No

1: Yes

#5 MPR Multi-spindle override effective mode

0: Preposition

1: Postposition

	#7	#6	#5	#4	#3	#2	#1	#0
3704								SSP

「Parameter type」: Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Power on

#0 SSP The Minimum Speed Unit of the Spindle

0: 1rpm

1: 0.1rpm

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

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00010000

「Valid mode」 : Immediate

#4 EVS For S Command when Using Spindle Control Function (spindle analog output or spindle serial output)

0: Do not output S code and SF

1: Output S code and SF

#7 CSGE Spindle S value of G00 command in constant surface speed mode

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
3706						MPA		

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#2 MPA In the Multi-Spindle Control, when the Spindle is Selected by the Address P, when the P Command is not Specified with the S Command

0: Alarm (PS5303)

1: Use the last P commanded by S_ P_; If P has not been commanded at all after power-on, use the value of parameter (No. 3775).

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

「Parameter type」 : Bit type

「Modify authority」: Equipment

「Parameter default」: 00000011

「Valid mode」: Immediate

#0 SAR Whether to Check the Spindle Speed Arrival Signal

0: Do not check

1: Check

#1 SAT When Starting to Execute the Thread Cutting Block, whether to Check the Spindle Speed Arrival Signal

0: Determine whether to check by the parameter SAR (NO.3708#0)

1: It has nothing to do with the parameter SAR, check

Note: When the thread cutting block is executed continuously, the thread cutting block after the second block does not check the spindle speed arrival signal.

#2 SSC Whether to Check the Spindle Speed during Cutting Feed

0: Do not check

1: Check

#6 TSO In the Thread Processing or Tapping Cycle, the Spindle Override

0: Invalid (fixed at 100%)

1: Valid

Note: In rigid tapping, the override is fixed at 100%, which has nothing to do with the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709						MSI		SAM

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000100

「Valid mode」: Immediate

#0 SAM The Number of Sampling when Seeking the Average Spindle Speed

0: 4 times (usually set to 0)

1: 1 time

#2 MSI In Multi-spindle Control, the SIND Signal is Valid

0: Only valid for the first spindle (the SIND signal of the second spindle is invalid)

1: Regardless of whether each spindle is selected, it is valid for each spindle (each spindle has its own SIND signal)

3710

CNC Controlled Spindle Number (CCS)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」 : 1~4

Set the number of spindles controlled by CNC.

	#7	#6	#5	#4	#3	#2	#1	#0
3713		MPC						

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#6 MPC In Multi-spindle Control, when the Spindle is Selected by the Program Command Based on the Address P, whether to Automatically Perform Thread Cutting/Feed per Revolution According to the Selected Spindle and Switch the Position Encoder Feedback Used

0: Don't switch

1: Switch

3715

Cs Switching Completion Delay Output Time Of Each Spindle (CST)

「Parameter type」 : Font spindle type

「Modify authority」 : Equipment

「Parameter default」 : 200

「Valid mode」 : Immediate

「Value range」 : 0~9999

This parameter sets the delay time (0~9999) ms for each spindle to output the completion signal when Cs is switched.

3717

Amplifier Number Of Each Spindle (NSS)

「Parameter type」 : Font spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」: -34~99

Set the amplifier number assigned to each spindle.

Parameter Setting Value	Corresponding Interface	Note
0	Spindle amplifier interface is not connected	
1~99	The logical ID number of the spindle connected via GSK-Link	Same as the setting value of the servo spindle logic ID number
-1~-4	Four groups of analog output ports corresponding to spindle interface 1 and spindle interface 2 on I/O unit 1	Use when using variable frequency spindle
-11~-14	Four groups of analog output ports corresponding to spindle interface 1 and spindle interface 2 on I/O unit 2	
-21~-24	Four groups of analog output ports corresponding to spindle interface 1 and spindle interface 2 on I/O unit 3	
-31~-34	Four groups of analog output ports corresponding to spindle interface 1 and spindle interface 2 on I/O unit 4	

3720

Number Of Encoder Lines For Each Spindle (CNT)

「Parameter type」: Font spindle type

「Modification authority」: Machine tool

「Parameter default」: 1024

「Valid mode」: Power on

「Value range」: 100~99999999

Set the number of lines of each spindle encoder.

3721

Number Of Teeth On One Side Of Each Spindle Position Encoder (GOE)

「Parameter type」: Font spindle type

「Modification authority」: Machine tool

「Parameter default」: 1

「Valid mode」: Power on

「Value range」: 1~9999

Set the number of teeth of each spindle position encoder side gear in speed control (feed per revolution, thread cutting, etc.).

3722

Number Of Teeth On One Side Gear Of Each Spindle (GOS)

「Parameter type」 : Font spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」 : 1~9999

Set the number of teeth of the gear on one side of each spindle during speed control (feed per revolution, thread cutting, etc.).

3723

Channel Number Corresponding To Each Spindle Encoder (CSE)

「Parameter type」 : Font spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : -31, -21, -11, -1, 0~2

Set the channel number corresponding to each spindle encoder.

Parameter Setting Value	Corresponding Interface	Note
0	The data of the spindle encoder comes from GSK-link transmission	Use when using GSK-Link spindle and without external encoder
1	Use the 1 st encoder channel interface	Use when using an external encoder
2	Use the 2 nd encoder channel interface	
-31, -21, -11, -1	Negative value means that the encoder value is obtained through the pulse I/O unit	

3730

Gain Adjustment Data Of Each Spindle Speed Analog Output(AGS)

「Parameter type」 : Font spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 1000

「Valid mode」 : Immediate

「Value range」 : 500~2000

「Data unit」 : 0.1%

Set the gain adjustment data of the spindle speed analog output.

Setting Method:

- (1) Set the standard setting value of 1000.
- (2) The spindle speed when the analog output maximum voltage of command spindle speed is 10V.
- (3) Measure the output voltage.
- (4) Set the value of the following formula on parameter No. 3730:
Set value = 10 (V) × 1000 / measured voltage (V)
- (5) After setting the parameters, again command the spindle speed analog output as the spindle speed of the maximum voltage, and confirm that the output voltage should be 10 volts.

3731

**Spindle Speed Analog Output Offset Voltage Compensation Value
(CVSS)**

「Parameter type」 : Font spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : -1000~1000

This parameter sets the compensation value of the spindle speed analog output bias voltage.

Setting Method:

- (1) Set the standard setting value 0.
- (2) Command to make the analog output voltage 0V theoretical spindle speed.
- (3) Measure the output voltage.
- (4) Set the value of the following formula on parameter No.3731:
Set value = $\frac{-8191 \times \text{Bias voltage (V)}}{12.5}$
- (5) After setting the parameters, command the theoretical spindle speed when the analog output voltage is 0V again, and confirm that the voltage is 0V.

3740

Time Of Spindle Speed Arrival Signal (SAD)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 3000

「Valid mode」 : Immediate

「Value range」 : 0~32767

Set the time delay from the execution of the S function to the detection of the spindle speed arrival signal.

3741	Maximum Speed Of Each Spindle In Gear 1 (MSG1)
3742	Maximum Speed Of Each Spindle In Gear 2 (MSG2)
3743	Maximum Speed Of Each Spindle In Gear 3 (MSG3)
3744	Maximum Speed Of Each Spindle In Gear 4 (MSG4)

「Parameter type」 : Font spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 6000

「Valid mode」 : Immediate

「Value range」 : 0~65536 r/min

The above 4 parameters respectively set the maximum speed of the spindle corresponding to the gear stage.

3750	Sampling Cycle Of Spindle Encoder During Thread Processing (MSC)
------	--

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~100

Set the sampling period of the spindle encoder during thread processing (0~100ms)

3751	Sampling Cycle Of Spindle Encoder During Feed Processing Per Revolution (RSC)
------	---

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 10

「Valid mode」 : Power on

「Value range」 : 0~100

Set the sampling period of the spindle encoder (0~100ms) during feed per revolution.

3770

The Axis Used As The Basis For Calculation In Constant Line Speed Control (ACS)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~number of control axes

This parameter sets the axis used as the basis for calculation during constant surface speed control.

Note: When setting 0, the default is X-axis. At this time, the P value commanded in the G96 block has no meaning for the constant surface speed control.

3771

Constant Line Speed Control Mode (G96) Minimum Spindle Speed (CFL)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 50

「Valid mode」 : Immediate

「Value range」 : 0~65536

This parameter sets the minimum speed of the spindle during constant surface speed control. In constant line speed control (G96), when the spindle speed is lower than the speed given by the parameter, it will be clamped at this speed.

3772

Maximum Speed Of Each Spindle (MSS)

「Parameter type」 : Font spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 6000

「Valid mode」 : Immediate

「Value range」 : 0~65536 r/min

This parameter sets the upper limit speed of each spindle. When the commanded spindle speed exceeds the spindle upper limit speed, or when the spindle speed exceeds the spindle upper limit speed after the spindle speed multiplier, the actual spindle speed is clamped at the upper limit speed set by this parameter.

Note 1: When using constant surface speed control, regardless of whether G96 or G97 is commanded, the spindle speed will be clamped by the highest spindle speed.

3773

Restricted Speed Of Each Spindle (LGS)

「Parameter type」 : Font spindle type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~65536

Set the maximum speed of each spindle when the signal G270 is valid. When the set value is 0, it is invalid (0~65536) r/min.

3775

The Default Spindle Selection P Command Value In Multiple Spindles (MPD)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」 : 1~4

When parameter MPP (NO.3703#3) =1 and parameter MPA (NO.3706#2) =1 in multi-spindle control, set the default P command value when S_P_ is not commanded once after power-on.

3776

Spindle Corresponding To S Command When In Multi-Spindle Control And No P Value Is Commanded (MPD)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~4

When in multi-spindle control and no P value is commanded, set the spindle corresponding to the S command (0~4).

3780

Servo Current Detection Value Of Each Spindle (SEC)

「Parameter type」 : Font spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~5000

Set the threshold value of each spindle servo current detection (0~5000)*0.1A, when the setting value is 0, it means no detection.

	#7	#6	#5	#4	#3	#2	#1	#0
3800	SRV							

「Parameter type」 : Bit spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Power on

#7 SRV In the Spindle Control Function Based on the Servo Motor, as the Servo Motor Spindle

0: Not used

1: Used

	#7	#6	#5	#4	#3	#2	#1	#0
3801	SRM							

「Parameter type」 : Bit spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Power on

#7 SRM The Working Mode of the Slave Control Axis and the Master Control Axis in the Spindle Synchronization Control Function based on the Servo Motor

0: Not the same

1: Same

3810	The Main Control Axis Number In The Spindle Synchronization Control Based On The Servo Motor (SSC)
-------------	---

「Parameter type」 : Font spindle type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~4

This parameter is used to set the axis number of the main control axis in the synchronous control of the spindle based on the servo motor. If this axis is used as the main control axis, the parameter is set to 0. (0~4)

3812

Phase Offset Value In Synchronization Control Of Each Spindle (SPPD)

「Parameter type」 : Word spindle type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 :

IS—B	IS—C
-99999.999~99999.999	-9999.9999~9999.9999

Set the phase offset value during spindle synchronization control by the servo motor.

3813

Phase Error Detection Width In Synchronous Control Of Each Spindle (SPPC)

「Parameter type」 : Word spindle type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 :

IS—B	IS—C
0~99.999	0~99.9999

Set the phase error detection width (0~99.9999) in the spindle synchronization control based on the servo motor. When it is set to 0, it means no detection, and only the slave axis needs to be set.

3815

Width Of Phase Check Completion In Synchronous Control Of Each Spindle (SPPE)

「Parameter type」 : Word spindle type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0.1

「Valid mode」 : Immediate

「Value range」 :

IS—B	IS—C
0~99.999	0~99.9999

Set the phase verification completion width (0~99.9999) in the spindle synchronization control based on the servo motor. When it is set to 0, it means no verification, and the master and slave axes need to be set.

3816

**Dedicated Acceleration/Deceleration Time Constant For Each Spindle
Speed Control (SST)**

「Parameter type」 : Font spindle type

「Modify authority」 : Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~5000

Set the dedicated acceleration/deceleration time constant (0~5000) when performing spindle speed control based on the servo motor.

3820

Spindle Orientation Position For Each Spindle (SOP)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0.0000

「Valid mode」 : Immediate

「Value range」 : 0~360.0000

Set the orientation position (0~360.0000 degrees) for spindle control based on the servo motor.

3821

Spindle Orientation Velocity For Each Spindle (SOV)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 1000

「Valid mode」 : Immediate

Set the orientation speed for spindle control based on the servo motor. Unit: 1deg/min

3822

**Spindle Orientation Acceleration/Deceleration Time For Each Spindle
(SOA)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~2000

Set the orientation acceleration/deceleration time (0~2000) ms for spindle control based on the servo motor.

	#7	#6	#5	#4	#3	#2	#1	#0
4900	FDTs							FLR

「Parameter type」: Bit spindle type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 FLR The Unit Set by Parameters 4911 and 4912 in the Spindle Speed Change Detection Function

0: In units of 1%

1: In units of 0.1%

#7 FDTs Spindle speed fluctuation detection validity

0: Invalid

1: Valid

4911	As The Spindle Reaches The Commanded Speed Allowable Rate (SSQ)
-------------	--

「Parameter type」: Font spindle type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 1~1000

This parameter is set in the spindle speed change detection function as the spindle speed allowable rate (q) when the spindle reaches the commanded speed.

4912	Spindle Rate Of Change That Does Not Issue a Spindle Speed Change Detection Alarm (SSR)
-------------	--

「Parameter type」: Font spindle type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 1~1000

This parameter sets the spindle speed change rate (r) that does not issue an alarm in the spindle speed change detection function.

4913	Spindle Speed Variation Range For Which The Spindle Speed Variation Detection Alarm Is Not Issued (SSI)
-------------	--

「Parameter type」: Font spindle type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 0~99999

This parameter sets the allowable variation range (i) that does not issue an alarm in the spindle speed variation detection function.

4914

The Time From The Change Of The Commanded Speed To The Start Of Spindle Speed Change Detection (SSP)

「Parameter type」: Font spindle type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 1~999999

This parameter sets the time (p) from when the commanded speed changes to the start of spindle speed change detection in the spindle speed change detection function.

1.16 Parameters related to tool compensation

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#6 EVO In the Tool Offset Compensation Mode, when the Compensation Amount is Changed

0: Valid from the next block of specified T code

1: Valid from the next buffer block

#7 TABR Measurement input mode

0: Normal mode

1: Mode with wear compensation

	#7	#6	#5	#4	#3	#2	#1	#0
5002		LWM		LGT	LGR			LD1

「Parameter type」: Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 LD1 The Offset Number of Tool Offset

0: Specify with the last 2 digits of the T code

1: Specify with the last digit of the T code

#3 LGR Tool Offset Execution Mode is the Tool Offset that is not Executed when the Coordinate Offset

0: Do not memorize

1: Memorize

#4 LGT Tool Offset Compensation Mode

0: Use coordinate system offset compensation (in this case, it has nothing to do with LWM, it is compensated in the block of T code)

1: Use tool movement compensation

#6 LWM When the Tool Offset Compensation Mode is Tool Movement (shape compensation and wear compensation when LGT is 1)

0: Execute compensation in T code block

1: Simultaneously with axis movement

Note: When LGT is 0, the offset is executed in the T code block, which has nothing to do with this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5003		LVC				CCN		

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#2 CCN G28 Cancels Tool Compensation Mode in the Tool Nose Radius Compensation Mode

0: Move to the middle point

1: Cancel only when moving to the reference point

#6 LVC Tool offset in non-MDI (Manual Data Input) mode

0: Do not clear at reset

1: Clear at reset

Note: The function of resetting and clearing tool offsets must be valid in non-MDI mode.

	#7	#6	#5	#4	#3	#2	#1	#0
5004					TS1		ORC	

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#1 ORC Set Value Of Tool Position Offset Amount

0: Compensate with diameter value

1: Compensate as radius value (This parameter is only valid for diameter-specified axes)

#3 TS1 The Measured Value of the Tool Compensation Amount is Directly Input into the Touch Detection of the Touch Sensor in the B Function

0: Through 4 contact inputs

1: Through 1 contact input

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 10000000

「Valid mode」: Immediate

#2 PRC In the Direct Input of the Compensation Amount of the Tool Offset, the Position Recording Function

0: Not used

1: Used

#5 QNI In the Direct Input Function B of the Measured Value of the Tool Compensation Amount, the Selection of the Tool Compensation Number

0: The operator selects the operation through the cursor

1: By inputting the signal from PLC

#7 TMS When Modifying the Current Tool Offset in the Reset State, whether to Update the Current Coordinates

0: Do not update

1: Update

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

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#1 TGC When T Code is Commanded in a Block Containing G50, G04 or G10

0: No alarm

1: P/S alarm

	#7	#6	#5	#4	#3	#2	#1	#0
5008		CNS	CNF	MCR	CNV		CNC	CNI

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 CNI Tool Nose Radius Compensation Interference Check

0: Execute

1: Do not execute

#1 CNC When the difference between the programmed movement direction and the offset movement direction is 90° ~270° in the interference check of radius compensation

0: P/S alarm

1: No alarm

#3 CNV Tool Nose Radius Compensation (T system) Interference Check and Vector Elimination

0: Execute

1: Do not execute

#4 MCR If G41/G42 Tool Nose Radius Compensation is Commanded in the MDI Mode, whether it will Give an Alarm

0: No alarm

1: P/S alarm

Note: In MDI mode, tool nose radius compensation is not performed, regardless of whether this parameter is set or not.

#5 CNF In the Interference Check of Tool Nose Radius Compensation, whether there is an Alarm when Cutting the Inside of a Full Circle

0: P/S alarm

1: No alarm

#6 CNS In the Interference Check of the Tool Nose Radius Compensation, whether there is an Alarm for the Steps Smaller than the Tool Radius

0: P/S alarm

1: No alarm

	#7	#6	#5	#4	#3	#2	#1	#0
5009				TSD				GSC

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 GSC Tool Compensation Measurement Value is Directly Input to the Offset Write Input Signal in Function B

0: Input from the mechanical side

1: Input from the PLC side

#4 TSD The Measured Value of the Tool Compensation Amount is Directly Input into the Function B, and the Movement Direction is judged as the Specification

0: Invalid

1: Valid

5010	In Tool Nose Compensation, The Limit Value Of The Vector Is Ignored When The Tool Moves Along The Outside Of The Corner (CLV)
------	---

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric input	0~9.999	0~9.9999	mm
Imperial input	0~0.9999	0~0.99999	inch

Set the limit value of the slight movement that is ignored when the tool moves along the outside of the corner in the tool nose radius compensation.

5011

The Maximum Error When There Is No Intersection In The Tool Nose Compensation (CRV)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input/Output」 : Input related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~9.999	0~9.9999	mm
Imperial input	0~0.9999	0~0.99999	inch

Set the maximum allowable error value in tool nose compensation when there is no intersection between two connecting segments.

5013

Maximum Tool Wear Compensation Amount (MTW)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 1.0

「Input/Output」 : Input related

「Valid mode」 : Immediate

「Value range」 :

		IS-B	IS-C
Predetermined area	Metric input(mm)	0~99999.999	0~9999.9999
	Imperial input(inch)	0~9999.9999	0~999.99999

This parameter sets the maximum value of tool wear compensation.

Note: When the absolute value of the set tool wear compensation exceeds this maximum value, the following alarm will appear:

Enter from Enter...Warning: Too many digits. Out of range (XXXX—XXXX) (input range in brackets)

Input with G10... An alarm occurs: The offset input with G10 exceeds the specified range.

5014

Maximum Incremental Tool Wear Compensation (MTS)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0.1

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

		IS-B	IS-C
Predetermined area	Metric input (mm)	0~99999.999	0~9999.9999
	Imperial input (inch)	0~9999.9999	0~999.99999

This parameter sets the maximum incremental value of tool wear compensation.

5015

During Manual Tool Measurement, The Distance To The X+ Contact Surface Of The Detection Sensor (X1P)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	-99999.999~99999.999	-9999.9999~9999.9999	mm
Imperial machine	-9999.9999~9999.9999	-999.99999~999.99999	inch
Rotary axis	-99999.999~99999.999	-9999.9999~9999.9999	deg

Set the record of the measurement reference position to each contact surface of the detection sensor. For the axis of diameter programming, the set value is specified by the diameter value.

5016

During Manual Tool Measurement, The Distance To The X-Contact Surface Of The Detection Sensor (X1M)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」：

Value Range	IS-B	IS-C	Unit
Metric machine	-99999.999~99999.999	-9999.9999~9999.9999	mm
Imperial machine	-9999.9999~9999.9999	-999.99999~999.99999	inch
Rotary axis	-99999.999~99999.999	-9999.9999~9999.9999	deg

Set the record of the measurement reference position to each contact surface of the detection sensor. For the axis of diameter programming, the set value is specified by the diameter value.

5017

During Manual Tool Measurement, The Distance To The Z+ Contact Surface Of The Detection Sensor (Z1P)

「Parameter type」：Font type-real number type

「Modify authority」：Equipment

「Parameter default」：0

「Input and output」：Output related

「Valid mode」：Immediate

「Value range」：

Value Range	IS-B	IS-C	Unit
Metric machine	-99999.999~99999.999	-9999.9999~9999.9999	mm
Imperial machine	-9999.9999~9999.9999	-999.99999~999.99999	inch
Rotary axis	-99999.999~99999.999	-9999.9999~9999.9999	deg

Set the record of the measurement reference position to each contact surface of the detection sensor. For the axis of diameter programming, the set value is specified by the diameter value.

5018

During Manual Tool Measurement, The Distance To The Z-Contact Surface Of The Detection Sensor (Z1M)

「Parameter type」：Font type-real number type

「Modify authority」：Equipment

「Parameter default」：0

「Input and output」：Output related

「Valid mode」：Immediate

「Value range」：

Value Range	IS-B	IS-C	Unit
Metric machine	-99999.999~99999.999	-9999.9999~9999.9999	mm
Imperial machine	-9999.9999~9999.9999	-999.99999~999.99999	inch
Rotary axis	-99999.999~99999.999	-9999.9999~9999.9999	deg

Set the record of the measurement reference position to each contact surface of the detection sensor. For the axis of diameter programming, the set value is specified by the diameter value.

5020

**The Measured Value Of The Tool Compensation Amount Is Directly
Input To The Tool Offset Number In Function B (TSB)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 1

「Valid mode」 : Immediate

「Value range」 : 1~99

Set the tool offset measurement value and directly input the tool offset number in function B (when setting the workpiece coordinate system offset value).

5021

**The Number Of Movement Interpolation Cycles Stored Before The
Detection Sensor Is Touched During Manual Tool Measurement**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~8

Set the number of movement interpolation cycles stored immediately before the detection sensor is touched. When set to 0, it is regarded as 8.

5022

The Feed Rate Of Each Axis During Manual Tool Measurement (HTSV)

「Parameter type」 : Word axis type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 100

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C
Metric machine	6~15000	6~12000
Imperial machine	6~12000	6~6000
Rotary axis	6~15000	6~12000

Set the feed rate of each axis during manual tool measurement, unit: 1mm/min or 1inch/min or

1deg/min.

5023

**Effective Distance Of Each Axis During Manual Tool Measurement
(HTSD)**

「Parameter type」 : Word axis type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C
Metric machine	0~99999.999	0~9999.9999
Imperial machine	0~9999.9999	0~999.99999
Rotary axis	0~99999.999	0~9999.9999

Set the effective distance of each axis during manual tool measurement. When it is set to 0, the effective distance will not be detected.

5025

**Set The Data Write Delay Time During Manual Tool Measurement
(HTDT)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~3000 ms

Set the data write delay time during manual tool measurement.

5043

Use The First Offset Axis Number (YNSA1)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~8

Set the axis number used to compensate the tool offset of the first offset axis, 0 means not used.

5044

Use The Axis Number Of The Second Offset Axis (YNSA2)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~8

Set the axis number used to compensate the tool offset of the second offset axis, 0 means not used.

5045

Use The Axis Number Of The Third Offset Axis (YNSA3)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~8

Set the axis number used to compensate the tool offset of the third offset axis, 0 means not used.

5046

Use The Axis Number Of The Fourth Offset Axis (YNSA4)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~8

Set the axis number used to compensate the tool offset of the fourth offset axis, 0 means not used.

5047

Use The Axis Number Of The Fifth Offset Axis (YNSA5)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~8

Set the axis number used to compensate the tool offset of the fifth offset axis, 0 means not used.

1.17 Parameters related to the canned cycle

The unit of canned cycle parameter setting is as follows.

Setting Unit	IS-B	IS-C	Unit
Metric input	0.001	0.0001	mm
Imperial input	0.0001	0.00001	inch

1.17.1 Parameters related to drilling canned cycles

	#7	#6	#5	#4	#3	#2	#1	#0
5101						RTR		DBX

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 DBX In drilling and boring commands, the drilling axis

0: Selected via plane selection

1: G83/G85 fixed as Z-axis, G87/G89 fixed as X-axis

#2 RTR In G83 and G87

0: Specify high-speed deep hole drilling cycle

1: Specify the deep hole drilling cycle

	#7	#6	#5	#4	#3	#2	#1	#0
5102	CSB					AUT	MRC	CTR

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 CTR In the Compound Cycle (G71, G72, G73), the Rough Cutting Target Shape

0: Not processed by C tool compensation

1: After C tool compensation

#1 MRC In the Compound Cycle (G71, G72, G73), whether to Check Monotonicity (this parameter is valid when parameter 5102.BIT2 is set to 1)

0: Do not check

1: Check

#2 AUT In multiple cycle commands (G71 or G72), machining is performed by automatically identifying Type I and Type II based on the monotonicity of the program

0: Valid

1: Invalid

#7 CSB In the Compound Cycle (G71, G72, G73), the Execution Mode of Single-stage Operation

0: Stop every cycle

1: Each line of the program stops

	#7	#6	#5	#4	#3	#2	#1	#0
5104						FCK		

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#2 FCK In the Compound Canned Cycle (G71, G72, G73), Processing the Shape

0: Do not check

1: Check

	#7	#6	#5	#4	#3	#2	#1	#0
5105	ESC					RF2	RF1	

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#1 RF1 In the Compound Canned Cycle G71, G72 Type I, whether to Perform Rough and Finishing Cutting

0: Proceed

1: Do not proceed

#2 RF2 In the Type II of Compound Canned Cycle G71, whether to Perform Rough and Finishing Cutting

0: Proceed

1: Do not proceed

#7 ESC In Compound Fixed Cycle G71 I Type Cycle Processing, whether to Retract the Tool Quickly

0: Normal retreat

1: Quick retreat

5110

M Code For Locking The C Axis In The Drilling Canned Cycle (CMD)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 35

「Valid mode」: Immediate

「Value range」: 3~97

In the drilling canned cycle, the M code of the C axis is locked.

5114

High-Speed Deep Hole Drilling Cycle Return Amount (HPDCRD)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0.1

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

This parameter sets the return value d of the G83, G87 high-speed deep hole drilling cycle.

5115

Pitch Amount Of Deep Hole Drilling Cycle (PDCRD)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0.1

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

This parameter sets the idle stroke amount d for G83 and G87 deep hole drilling cycles.

1.17.2 Parameters related to thread cutting cycle

5128	Detection Distance When Executing Thread Cutting Run-Out Action (TZD)
------	---

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0.1

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric Input	0~99.999	0~99.9999	mm
Imperial input	0~9.9999	0~9.99999	inch

Set the actual detection distance value (0~9.9999) when the thread cutting run-out long axis run-out distance is zero.

5129	Thread Cutting Cycle G92, G76 Mode (THM)
------	--

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 1

「Valid mode」: Immediate

「Value range」: 0~1

Set the thread cutting cycle G92, G76 mode.

0: Normal mode 1: High-speed mode

5130	Chamfer Amount (THD) For Thread Cutting Cycles (G76, G78, G92)
------	--

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~99×(0.1 pitch)

This parameter sets the chamfer amount for thread cutting cycles G76, G78, and G92. Range: 0~99×(0.1 pitch).

5131

The Run-Out Angle Of Thread Cutting Cycle G92, G78, G76 (CAT)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~89

This parameter sets the thread run-out angle in the thread cutting cycle (G76, G78, G92). When the setting value is 0, it is equivalent to setting an angle of 45°.

1.17.3 Parameters related to the compound canned cycle

5132

Cutting-In Amount Of Compound Canned Cycle G71, G72 (THC)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0.1

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the cutting amount of compound canned cycle G71 and G72.

5133

Retraction Amount Of Compound Canned Cycle G71, G72 (MCE)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

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

5134

Clearance Amount For Compound Fixed Cycles G71 And G72 (AOC)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

Set the clearance amount for compound fixed cycles G71 and G72 (0~9999.9999)

5135	Retraction Amount Of Compound Canned Cycle G73 Along The X-Axis (G73XE)
5136	Retraction Amount Of Compound Canned Cycle G73 Along The Z-Axis (G73ZE)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric input	-99999.999~99999.999	-9999.9999~9999.9999	mm
Imperial input	-9999.9999~9999.9999	-999.99999~999.99999	inch

Set the retraction amount of the compound canned cycle G73 along the X and Z axes.

5137	The Number Of Divisions Of The Compound Canned Cycle G73 (G73DC)
------	---

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 1

「Valid mode」: Immediate

「Value range」: 1~999

Set the number of divisions of the compound canned cycle G73 (1~999).

5139	The Retreat Amount Of Compound Canned Cycle G74, G75 (G74G75R)
------	---

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0.1

「Input/Output」: Input related

「Valid mode」 : Immediate

「Value range」 : 0~9999.9999

Set the retraction amount of compound canned cycle G74 and G75.

5140

Minimum Cutting Amount Of Compound Canned Cycle G76 (G76MID)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input/Output」 : Input related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the minimum cutting amount of the compound canned cycle G76.

5141

Finishing Allowance Of Compound Canned Cycle G76 (G76FA)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0.05

「Input/Output」 : Input related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the finishing allowance of the compound canned cycle G76.

5142

Compound Canned Cycle G76 Finishing Cycle Number (G76FC)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 1

「Valid mode」 : Immediate

「Value range」 : 1~99

Set compound canned cycle G76 finishing cycle times.

5143

Compound Canned Cycle G76 Tool Nose Angle (G76TNA)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 60

「Valid mode」 : Immediate

「Value range」 : 0~99 (deg)

Set compound canned cycle G76 tool nose angle.

5144

Cutting Mode Of G78 Tool Edge In Compound Canned Cycle (G78TCI)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 2

「Valid mode」 : Immediate

「Value range」 : 0~3

Set the cutting mode of the compound canned cycle G78 tool edge.

0: Along the center line of the thread profile.

1: Along the left side of the thread profile.

2: Along the right side of the thread profile.

3: Cut in turns along the thread profile left and right (0~3).

5145

The Depth Of Cut Method Of Compound Canned Cycle G78 Feed (G78TCD)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 1

「Valid mode」 : Immediate

「Value range」 : 0~1

Set the cutting depth mode of the compound canned cycle G78 feed.

0: Equidistant infeed.

1: Decreasing feed (decreasing feed is the same as G76 feed) (0~1).

5149

Retraction Action Override Of Boring Cycle (G85, G89)(BCRDOV)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 200

「Valid mode」 : Immediate

「Value range」 : 0~2000

Set the speed override value (%) during the retreat action in the boring cycle, which is independent of the feedrate override. When set to 0, it is equivalent to setting the speed override of 200%.

5150

Short-axis feed amount (D) for thread tapping and retracting cutting cycle (G92.2)(XIT)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input/Output」 : Input related+

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the precession of the minor axis of the precession and recirculation cutting cycle (G92.2).

5151

Long-axis feed amount (E) for thread tapping and retracting cutting cycle (G92.2)(ZIT)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input/Output」 : Input related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the major axis feed amount for the threading in/out cutting cycle (G92.2).

5152

Thread Screw In And Screw Out Cutting Cycle (G92.2) Minor Axis Screw Out (J) (XOT)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」: 0

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the minor axis screw-out amount for the in/out cutting cycle (G92.2).

5153

Major Axis Screw-Out Amount (K) For Thread In/Out Cutting Cycle (G92.2) (ZOT)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the major axis screw-out amount for the in/out cutting cycle (G92.2).

1.18 Parameters related to rigid tapping

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 10000100

「Valid mode」: Immediate

#0 G84 Method for commanding rigid tapping

0: Command the M code for rigid tapping before commanding G84/G88 (see parameter No.5210)

1: Do not use M code to command rigid tapping; G84/G88 are used as G codes for rigid tapping, not for ordinary tapping

- #2 CRG** Whether to wait for the rigid tapping signal RGTAP to become 0 when commanding the rigid tapping release instruction
 0: Do not wait
 1: Wait
- #4 DOV** Feed rate override during pull-out operation in rigid tapping
 0: Invalid
 1: Valid (override value is set in parameter No.5211)
- #5 PCP** When address Q is commanded in tapping cycle/rigid tapping
 0: Used as high-speed deep hole tapping cycle
 1: Used as deep hole tapping cycle
- #6 FHD** Feed hold and single block operation during rigid tapping
 0: Not allowed
 1: Allowed
- #7 SRS** Spindle selection for rigid tapping in multi-spindle control
 0: Use spindle selection signals SWS1~SWS3
 1: Use rigid tapping spindle selection signals RGTSP1~RGTSP3

	#7	#6	#5	#4	#3	#2	#1	#0
5201				OV3	OVU	TDR		

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

- #2 TDR** In Rigid Tapping, Cutting Time Constant
 0: Use the same parameter No.5261 when infeed and drawing
 1: Use different parameters when infeed and drawing, use parameter No.5261 when infeed, and use No.5271 when drawing
- #3 OVU** Set the Unit of the Parameter (No.5211) of the Drawing Magnification of Rigid Tapping
 0: Set to 1%
 1: Set to 10%
- #4 OV3** Command the spindle speed during pull-out via the program (address J), thereby enabling the override during the pull-out operation
 0: Invalid
 1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
5202		OVE						

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#6 OVE The command range for the pull-out override command (address J) based on the rigid tapping program instruction is:

0:100%~200%

1:100%~2000%

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS				

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#4 OVS In rigid tapping, the override based on the feed rate override selection signal and the override cancel signal

0: Invalid

1: Valid

Note 1: When the feed rate override is enabled, the pull-out override becomes invalid.

Note 2: The spindle override is fixed at 100% during rigid tapping, which is independent of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5209								RTX

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000001

「Valid mode」: Immediate

#0 RTX In Rigid Tapping, Drilling Axis

0: Select by plane selection

1: G84 is fixed as Z-axis and G88 is fixed as X-axis

5210**M Code For Instructing Rigid Tapping (RTMC)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~255

Set the M code that specifies the rigid tapping method. When it is set to 0, CNC considers it as M29.

5211**The Magnification During The Drawing Action Of Rigid Tapping (RTEOV)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~200

「Data Unit」 : 1% or 10% (decided by No.5201#3)

Set the magnification value during the pulling action of rigid tapping.

Note 1: The override value is valid when parameter No.5200#4 (DOV) is set to "1".

Note 2: When parameter No.5201#3 (OVU) is set to "1", the unit of the set data becomes 10%, and the override can be applied for pull-out operations up to 2000%.

5213**Deep Hole Rigid Tapping Return Amount (PRTRD)**

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input/Output」 : Input related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the retraction amount of the high-speed deep hole tapping cycle or the empty stroke amount of the deep hole tapping cycle.

5241**Maximum Spindle Speed During Rigid Tapping (RTMS)**

「Parameter type」: Font spindle type

「Modify authority」: Equipment

「Parameter default」: 2000

「Valid mode」: Immediate

「Value range」: 0~9999

Set the maximum spindle speed for rigid tapping.

5261

Rigid Tapping Acceleration And Deceleration Time Constant (RTLT)

「Parameter type」: Font spindle type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 0~4000 ms

Set the linear acceleration/deceleration time constant of the spindle and tapping axis in rigid tapping.

5271

Acceleration And Deceleration Time Constant During Rigid Tapping And Drawing (RTET)

「Parameter type」: Font spindle type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 0~4000 ms

Set the linear acceleration/deceleration time constant of the main axis and the tapping axis during the drawing operation in rigid tapping.

Note: When parameter No.5201#2 (TDR) is "1", this parameter is valid.

5275

The Actual Movement Of The Tapping Axis In G84/G88 Lags The Number Of Compensation Cycles Sampled By The Spindle Encoder (ZBK)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 6

「Valid mode」: Immediate

「Value range」: 0~10

Set in G84/G88 ordinary tapping (non-rigid tapping), the movement of the tapping axis lags the number

of compensation cycles sampled by the spindle encoder, generally set to 4~8.

5276

Spindle Detection Speed In Ordinary Tapping (OTCT)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~100

Set the spindle speed value detected when the normal attack speed is retracted, and the unit is 0.1 rpm.

5282

Return Amount During Rigid Tapping Return (OVRTT)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input/Output」: Input related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric input	0~99999.999	0~9999.9999	mm
Imperial input	0~9999.9999	0~999.99999	inch

Set the excess return amount in rigid tapping return, that is, increase the value set by this parameter (0~9999.9999) after returning to point R.

1.19 Parameters related to special processing

5310

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

								LFSM
--	--	--	--	--	--	--	--	------

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 LFSM Operating mode for chip breaking cutting G01

0: Mode A

1: Mode B

5311

Frequency Value Of Chip-Breaking Cutting Retreat (LSKF)

「Parameter type」 : Font type
 「Modify authority」 : Equipment
 「Parameter default」 : 1
 「Valid mode」 : Immediate
 「Value range」 : 1~10

Set the frequency value (1~10) of the chip-breaking cutting retraction function.

5312

The Amplitude Value Of Chip Breaking Cutting Retreat (LSKR)

「Parameter type」 : Font type-real number type
 「Modify authority」 : Equipment
 「Parameter default」 : 0.02
 「Input/Output」 : Input related
 「Valid mode」 : Immediate
 「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	0~9.999	0~9.9999	mm
Imperial input	0~0.9999	0~0.99999	inch

Set the amplitude value (1~9.9999) of the chip-breaking cutting retraction function.

5320

Spindle Encoder Associated With Each Axis (FSSE)

「Parameter type」 : Word axis type
 「Modify authority」 : Equipment
 「Parameter default」 : 0
 「Valid mode」 : Immediate

Set the spindle encoder (0~4) corresponding to each axis in feed per revolution.

5330

Set The Table Movement Axis (WBA)

「Parameter type」 : Font type
 「Modify authority」 : Equipment
 「Parameter default」 : 0
 「Valid mode」 : Immediate

Set the axis number (0~8) for the table movement axis.

5331

Set The Lifting Platform Movement Axis (LPA)

「Parameter type」 : Font type
 「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

Set the axis number (0~8) for the lifting platform movement axis.

5332

Set The Spindle Corresponding To The Graphite Wheel (GWA)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

Set the spindle number (0~4) corresponding to the graphite wheel.

5333

Set The Graphite Wheel Following Spindle (GWS)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

Set the spindle number (0~4) that the graphite wheel follows.

5334

Set The Graphite Wheel Diameter (GWD)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

Set the diameter value of the graphite wheel (0~9999.9999).

5360

**Specification Of The Rotary Axis In G6.4, G6.5, And G6.6 Commands
(EIC)**

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 3

「Valid mode」 : Immediate

「Value range」 : 1~number of control axes

Set the control axis number (1 to number of control axes) of the rotary axis when interpolating G6.4, G6.5, and G6.6 commands.

5361

**Specification Of The Linear Axis In G6.4, G6.5, And G6.6 Commands
(EIL)**

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 1

「Valid mode」: Immediate

「Value range」: 1~8

Set the control axis number (1 to number of control axes) of the linear axis when interpolating G6.4, G6.5, and G6.6 commands.

1.20 Parameters related to polar coordinate interpolation

	#7	#6	#5	#4	#3	#2	#1	#0
5400							RSH	

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Power on

#1 RSH Whether to Display the Rotating Coordinate Page

0: Do not display

1: Display

5410	Rotation Angle Used When There Is No Rotation Angle Command In Coordinate Rotation(ANG)							
------	--	--	--	--	--	--	--	--

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: -360.0000~360.0000

Set the rotation angle used when there is no rotation angle command in the coordinate rotation.

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

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#1 AFC Whether to perform automatic override and automatic speed clamping in polar coordinate interpolation mode

0: Not performed

1: Performed

Note: In the polar coordinate interpolation mode, the closer the tool is to the center of the workpiece, the greater the speed component of the rotary axis, and the center part may exceed the maximum cutting speed (parameter 5462). At this time, a servo alarm (NO.411) will appear. The automatic feedrate override and automatic feedrate clamp function will automatically control the feedrate so that the speed component of the rotary axis does not exceed the maximum cutting feed.

#6 PRD When the linear axis in polar coordinate interpolation mode is specified as diameter

0: Diameter value

1: Radius value

5460

Specification Of Polar Coordinate Interpolation Axis (Linear Axis) (LAI)

5461

Specification of Polar Coordinate Interpolation Axis (Rotary Axis) (RAI)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Valid mode」: Immediate

「Value range」: 1~number of control axes

「Factory default」: NO.5460 is 1; NO.5461 is 5

Set the control axis number of the linear axis and rotary axis for polar coordinate interpolation.

5462

Maximum Cutting Feed Rate For Polar Coordinate Interpolation (MFI)

「Parameter type」: Font type-real number type

「Modification authority」: Machine tool

「Parameter default」: 8000

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

	IS-B	IS-C	Unit
Metric machine	0, 6~100000	0, 6~60000	mm/min
Imperial machine	0, 6~48000	0, 6~24000	inch/min

Set the upper limit of the effective feed rate for polar coordinate interpolation. If the commanded speed

is greater than this value, it will be clamped to this speed. When this parameter is set to 0, the polar coordinate interpolation will be clamped using the value of the normal maximum cutting feed rate (parameter 1422).

5463

Percentage Of Automatic Override Allowed In Polar Coordinate Interpolation (API)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~100

When setting polar coordinate interpolation, in order to limit the percentage of the automatic override allowed by the cutting feedrate of the rotary axis.

Allowable speed of the rotary axis

= Maximum cutting feed rate × percentage of override

In the polar coordinate interpolation mode, the closer the tool is to the workpiece center, the greater the speed component of the rotary axis. When the allowable speed is exceeded, the feed speed is automatically multiplied by the magnification value calculated by the following formula:

Multiplying factor = allowable speed of the rotary axis / speed component of the rotary axis × 100%

If the rotation speed multiplied by the magnification is still greater than the allowable speed, the feed speed is clamped to the maximum allowable cutting feed speed (automatic speed clamp function).

Note: When the value of this parameter is set to 0, it is regarded as 90%; in order to perform automatic speed override and automatic speed clamping, parameter 5450#1 (AFC) must be set to 1.

5464

Compensation Amount Of Imaginary-Axis Direction Error In Polar Coordinate Interpolation Method (LAC)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Input/Output」 : Input related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric input	-99.999~99.999	-99.9999~99.9999	mm
Imperial input	-9.9999~9.9999	-9.99999~9.99999	inch

When the center of the rotation axis of the polar coordinate interpolation is not on the linear axis, set the error value (-99.9999~99.9999).

1.21 Parameters about indexing table

	#7	#6	#5	#4	#3	#2	#1	#0
5500	IDX						ISP	

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#1 ISP Servo shutdown of the indexing axis when clamping is completed

0: Processing on the CNC side

1: No processing on the CNC side (depending on the state of the G126 servo off signal)

#7 IDX The Action Sequence of Indexing Table Indexing Function (valid when parameter 5500#1 is 0)

0: Type A

1: Type B

By setting the parameter ITI, the indexing function of the indexing table can be temporarily disabled. When the setting is invalid, manual, MPG, increment and zero return operations can be performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5510	ICAx							

「Parameter type」 : Bit axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

#7 ICAX Whether the indexing table control axis function of the corresponding axis is valid

0: Invalid

1: Valid

5520

Indexing Table Signal Detection Time (TAT)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 5000

「Valid mode」: Immediate

「Value range」: 0~10000

Set the detection time (0~10000) ms for the indexing table's unclamping and clamping signals.

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

5650

FGRS

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#7 FGRS Gear ratio selection for flexible synchronous control mode

0: Parameter 5680

1: Parameter 5690

5660

Master Axis Number For Flexible Synchronous Control (FMAU)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 5000

「Valid mode」: Immediate

「Value range」: 0~8

Set the master axis number for flexible synchronous control. A setting of 0 indicates ineffectiveness (0~8).

5661

Slave Axis Number For Flexible Synchronous Control (FSAU)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」 : 0~8

Set the slave axis number for flexible synchronous control. A setting of 0 indicates ineffectiveness (0~8).

5670

M Code For Enabling Flexible Synchronous Control Mode (FMON)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 45

「Valid mode」 : Immediate

「Value range」 : 1~200

Set the M code for enabling flexible synchronous control mode (1~200).

5671

M Code For Disabling Flexible Synchronous Control Mode (FMOF)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 46

「Valid mode」 : Immediate

「Value range」 : 1~200

Set the M code for disabling flexible synchronous control mode (1~200).

5680

Numerator Of Gear Ratio For Flexible Synchronous Control Mode (FNGR)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 1

「Valid mode」 : Immediate

「Value range」 : -99999999~99999999

Set the numerator of the gear ratio for flexible synchronous control mode (-99999999~99999999)

5681

Denominator Of Gear Ratio For Flexible Synchronous Control Mode (FDGR)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 1

「Valid mode」 : Immediate

「Value range」 : -99999999~99999999

Set the denominator of the gear ratio for flexible synchronous control mode (-99999999~99999999)

5690

Outer Diameter Before Machining In Flexible Synchronous Control Mode (FEBW)

- 「Parameter type」 : Font type
- 「Modify authority」 : Equipment
- 「Parameter default」 : 1
- 「Valid mode」 : Immediate
- 「Value range」 : 0~9999.9999

Set the outer diameter before machining in flexible synchronous control mode (0~9999.9999)

5691

Wall Thickness Before Machining In Flexible Synchronous Control Mode (FTBW)

- 「Parameter type」 : Font type
- 「Modify authority」 : Equipment
- 「Parameter default」 : 1
- 「Valid mode」 : Immediate
- 「Value range」 : 0~9999.9999

Set the wall thickness before machining in flexible synchronous control mode (0~9999.9999)

5692

Outer Diameter After Machining In Flexible Synchronous Control Mode (FEAW)

- 「Parameter type」 : Font type
- 「Modify authority」 : Equipment
- 「Parameter default」 : 1
- 「Valid mode」 : Immediate
- 「Value range」 : 0~9999.9999

Set the outer diameter after machining in flexible synchronous control mode (0~9999.9999)

5693

Wall Thickness After Machining In Flexible Synchronous Control Mode (FTAW)

- 「Parameter type」 : Font type
- 「Modify authority」 : Equipment
- 「Parameter default」 : 1
- 「Valid mode」 : Immediate
- 「Value range」 : 0~9999.9999

Set the wall thickness after machining in flexible synchronous control mode (0~9999.9999)

	#7	#6	#5	#4	#3	#2	#1	#0
5700								LCF

「Parameter type」 : Bit type

「Modify authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

#0 LCF Whether the straightness error compensation function is valid.

0: Invalid

1: Valid

5710	Axis Number Of The 1st Group Movement Axis For Straightness Error Compensation (MAOL1)
------	---

「Parameter type」 : Font type

「Modify authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 :0~8

Set the axis number of the 1st group movement axis for straightness error compensation. A setting of 0 indicates ineffectiveness (0~8).

5711	Axis Number Of The 2nd Group Movement Axis For Straightness Error Compensation (MAOL2)
------	---

「Parameter type」 : Font type

「Modify authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 :0~8

Set the axis number of the 2nd group movement axis for straightness error compensation. A setting of 0 indicates ineffectiveness (0~8).

5720	Axis Number Of The 1st Group Compensation Axis For Straightness Error Compensation (CAOL1)
------	---

「Parameter type」 : Font type

「Modify authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~8

Set the axis number of the 1st group compensation axis for straightness error compensation. A setting of 0 indicates ineffectiveness (0~8).

5721

Axis Number Of The 2nd Group Compensation Axis For Straightness Error Compensation (CAOL2)

「Parameter type」 : Font type

「Modify authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~8

Set the axis number of the 2nd group compensation axis for straightness error compensation. A setting of 0 indicates ineffectiveness (0~8).

5730

Multiplier Of The 1st Group For Straightness Error Compensation (LCR1)

「Parameter type」 : Font type

「Modify authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」 : -100~100

Set the multiplier of the 1st group for straightness error compensation (-100~100).

5731

Multiplier Of The 2nd Group For Straightness Error Compensation (LCR2)

「Parameter type」 : Font type

「Modify authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」 : -100~100

Set the multiplier of the 2nd group for straightness error compensation (-100~100).

5740

Interval Of The 1st Group For Straightness Error Compensation (LCD1)

「Parameter type」 : Font type

「Modify authority」 : Machine tool

「Parameter default」 : 10

「Valid mode」 : Power on

「Value range」 : 1~1000

Set the interval of the 1st group for straightness error compensation (1~1000) mm.

5741

**Interval Of The 2nd Group For Straightness Error Compensation
(LCD2)**

「Parameter type」 : Font type

「Modify authority」 : Machine tool

「Parameter default」 : 10

「Valid mode」 : Power on

「Value range」 : 1~1000

Set the interval of the 2nd group for straightness error compensation (1~1000) mm.

5750

**Quantity Of The 1st Group For Straightness Error Compensation
(LCN1)**

「Parameter type」 : Font type

「Modify authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」 : 1~200

Set the quantity of the 1st group for straightness error compensation (1~200).

5751

**Quantity Of The 2nd Group For Straightness Error Compensation
(LCN2)**

「Parameter type」 : Font type

「Modify authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Power on

「Value range」 : 1~200

Set the quantity of the 2nd group for straightness error compensation (1~200).

5760

**Starting Coordinate Of The 1st Group For Straightness Error
Compensation (LCC1)**

「Parameter type」: Font type

「Modify authority」: Machine tool

「Parameter default」: 0

「Valid mode」: Power on

「Value range」: -5000~5000

Set the starting coordinate of the 1st group for straightness error compensation (-5000~5000) mm.

5761

Starting Coordinate Of The 2nd Group For Straightness Error Compensation (LCC2)

「Parameter type」: Font type

「Modify authority」: Machine tool

「Parameter default」: 0

「Valid mode」: Power on

「Value range」: -5000~5000

Set the starting coordinate of the 2nd group for straightness error compensation (-5000~5000) mm.

1.22 Parameters related to user macro program

6000

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

EMA SBM G67

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 G67 When the modal call (G66) mode of the macro program is not established, but the cancel modal call command (G67) is instructed.

0: P/S alarm occurs (No. 122)

1: G67 ignored

#5 SBM In the User Macro Program, whether to Use Single Block Stop

0: Not used

1: Used

#7 EMA G65 macro program function selection

0: G65 macro program call

1: Type A macro program

6001

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

CLV CCV

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 01000000

「Valid mode」 : Immediate

#6 CCV User Macro Program Public Variables 100~199, after Resetting in Non-logging Mode

0: Clear to empty value

1: Do not clear

Note: In the MDI mode, pressing reset cannot clear the macro program public variables.

#7 CLV User Macro Program Local Variables 1 ~ 33, after Resetting in Non-logging Mode

0: Clear to empty value

1: Do not clear

	#7	#6	#5	#4	#3	#2	#1	#0
6004	EDP						MFZ	NAT

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 NAT User Macro Program Function Commands ATAN and ASIN

0: The result of ATAN is 0~360.0 The result of ASIN is 270.0~0~90.0

1: The result of ATAN is -180.0~0~180.0 The result of ASIN is -90~0~90

#1 MFZ The Angle of the Operation Command SIN, COS or TAN of the User

1.0×10^{-8} Macro Program is or Smaller, or the Operation Result is not Accurate

Zero Time

0: Underflow processing

1: Reduced to 0

#7 EDP Precision setting for macro comparison operators

0: Invalid

1: Valid (set parameter 6010)

	#7	#6	#5	#4	#3	#2	#1	#0
6008	GMP	MMP	TMP					FOC

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000001

「Valid mode」 : Immediate

#0 F0C Macro Variable Operation Result

0: Alarm when the data range exceeds $\pm 1E308$

1: Alarm when the data range exceeds $\pm 1E47$

#5 TMP Allow T Code to Call Macro Program or not

0: Do not allow

1: Allow

#6 MMP Allow M code to call macro programs or not

0: Do not allow

1: Allow

#7 GMP Allow G code to call macro program or not

0: Do not allow

1: Allow

6010

Number Of Precision Digits For Values Compared With Comparison Operators (NPC)

「Parameter type」 : Font type

「Modification authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~5

Set the number of decimal places for the two values being compared when using the comparison operators in the macro program (0~5).

6020

Main Program Number Of Custom Station Program (CMM)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 9000

「Valid mode」 : Power on

「Value range」 : 9000~9999

Set the main program number in the custom station program. When the parameter 8133#7 is 1, the system will only load the main program set by this parameter.

6021

The Macro Variable Number Corresponding To The Subroutine Of The Custom Station Program (CMS1)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 500

「Valid mode」 : Power on

「Value range」 : 500~999

Set the macro variable number corresponding to the custom station 1 subroutine.

6022

The Macro Variable Number Corresponding To The Subroutine Of The Custom Station Program (CMS2)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 501

「Valid mode」 : Power on

「Value range」 : 500~999

Set the macro variable number (500~999) corresponding to the 2 subroutine in the custom station.

6031

The Start Number Of The Variable That You Want To Protect Among Public Variables (#500~#999) (MPH)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0, 500~999

Protect the variables (500 to 999) within the range set in the public variables (#500~#999).

6032

The Last Number Of The Variable That You Want To Protect Among Public Variables (#500~#999) (MPT)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0, 500~999

Protect the variables (500 to 999) within the range set in the public variables (#500~#999).

6060

Call The T Code Of The Program Number 9010 User Macro Program (TLM1)

6061	Call The T Code Of The Program Number 9011 User Macro Program (TLM2)
6062	Call The T Code Of The Program Number 9012 User Macro Program (TLM3)
6063	Call The T Code Of The Program Number 9013 User Macro Program (TLM4)
6064	Call The T Code Of The Program Number 9014 User Macro Program (TLM5)
6065	Call The T Code Of The Program Number 9015 User Macro Program (TLM6)
6066	Call The T Code Of The Program Number 9016 User Macro Program (TLM7)
6067	Call The T Code Of The Program Number 9017 User Macro Program (TLM8)
6068	Call The T Code Of The Program Number 9018 User Macro Program (TLM9)
6069	Call The T Code Of The Program Number 9019 User Macro Program (TLM10)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~9999

Specify the T code for calling the program number 9010~9019 user macro program.

6080	Call The M Code Of The Program Number 9020 User Macro Program (MLM1)
6081	Call The M Code Of The Program Number 9021 User Macro Program (MLM2)
6082	Call The M Code Of The Program Number 9022 User Macro Program (MLM3)
6083	Call The M Code Of The Program Number 9023 User Macro Program (MLM4)
6084	Call The M Code Of The Program Number 9024 User Macro Program (MLM5)
6085	Call The M Code Of The Program Number 9025 User Macro Program (MLM6)
6086	Call The M Code Of The Program Number 9026 User Macro Program (MLM7)
6087	Call The M Code Of The Program Number 9027 User Macro Program (MLM8)
6088	Call The M Code Of The Program Number 9028 User Macro Program (MLM9)
6089	Call The M Code Of The Program Number 9029 User Macro Program (MLM10)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0, 3~9999

Specify the M code for calling the program number 9020~9029 user macro program.

6090	Call The G Code Of The Program Number 9030 User Macro Program (GLM1)
6091	Call The G Code Of The Program Number 9031 User Macro Program (GLM2)
6092	Call The G Code Of The Program Number 9032 User Macro Program (GLM3)
6093	Call The G Code Of The Program Number 9033 User Macro Program (GLM4)
6094	Call The G Code Of The Program Number 9034 User Macro Program (GLM5)
6095	Call The G Code Of The Program Number 9035 User Macro Program (GLM6)
6096	Call The G Code Of The Program Number 9036 User Macro Program (GLM7)
6097	Call The G Code Of The Program Number 9037 User Macro Program (GLM8)
6098	Call The G Code Of The Program Number 9038 User Macro Program (GLM9)
6099	Call The G Code Of The Program Number 9039 User Macro Program (GLM10)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0, 500~599

This parameter is used to set the G code (500~599) of the user macro program that calls the program number.

1.23 Parameters related to jump function

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

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 TSG Detection Skip Signal, Multi-step Skip Signal SKIP, SKIP2~SKIP4 Type

0: Level signal

1: Rising edge or falling edge signal

#1 SK0 Set the Effective State of the Skip Signal

0: Valid when the input signal is "1"

1: Valid when the input signal is "0"

#7 SKF Dry Run and Override Jump Command to G31

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
6210		MDC						

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#6 MDC Will Be the Tool Measurement Value of Automatic Tool Compensation

0: Add to the current offset

1: Subtract from the current offset

	#7	#6	#5	#4	#3	#2	#1	#0
6240	IGA							AE0

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Power on

#0 AE0 Automatic Tool Compensation Signal XAE1 (X0.6), XAE2 (X0.7)

0: When it is 1, it is considered that the measurement position has been reached

1: When it is 0, it is deemed that the measurement position has been reached

#7 IGA Whether to Use Automatic Tool Compensation Function

0: Used

1: Not used

6241	Measure The Feed Rate During Automatic Tool Compensation (Used For Xae1 Signal) (ATOF1)
6242	Measure The Feed Rate During Automatic Tool Compensation (Used For Xae2 Signal) (ATOF2)

「Parameter type」: Font type-real number type

「Modification authority」: Machine tool

「Parameter default」: 1000

「Valid mode」: Immediate

「Input and output」: Output related

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	1mm/min	0, 6~15000	0, 6~12000	1000
Imperial machine	0.1inch/min	0, 6~6000	0, 6~4800	

These two parameters set the feed rate when measuring automatic tool compensation.

Note: When the setting value of parameter 6242 is 0, the setting value of parameter 6241 is valid.

6251	γ Value Of X-Axis In Automatic Tool Compensation (ATOR1)
6252	γ Value Of Z-Axis In Automatic Tool Compensation (ATOR2)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0.1

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	mm/min	0~99999.999	0~9999.9999	0.1
Imperial machine	inch/min	0~9999.9999	0~999.99999	

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

Note: Regardless of whether it is a diameter or a radius specification, the radius would always be used for setting.

6254

ϵ Value Of X-Axis In Automatic Tool Compensation (ATOE1)

6255

ϵ Value Of Z-Axis In Automatic Tool Compensation (ATOE2)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0.1

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Setting Unit	Data Unit	Valid Range		Factory Default
		IS-B	IS-C	
Metric machine	mm/min	0~99999.999	0~9999.9999	0.1
Imperial machine	inch/min	0~9999.9999	0~999.99999	

These two parameters sequentially set the ϵ in the automatic tool compensation function.

Note: Regardless of whether it is a diameter or a radius specification, the radius would always be used for setting.

6280

Torque Percentage For Torque Limit Skip In G31 Command (TPV)

「Parameter type」 : Font type

「Modification authority」 : Equipment

「Parameter default」 : 50

「Valid mode」 : Immediate

「Value range」 : 1~300

Set the torque percentage for torque limit skip in G31 command (1~300).

6281

Detection Delay Time For Torque Limit Skip In G31 Command (TPT)

「Parameter type」 : Font type

「Modification authority」 : Equipment

「Parameter default」 : 0

「Valid mode」: Immediate

「Value range」: 0~2000

Set the detection delay time for torque limit skip in G31 command (0~2000) ms.

1.24 Parameters related to MPG rollback

	#7	#6	#5	#4	#3	#2	#1	#0
6400		MGO				MC6	MC4	RPO

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 RPO In the MPG Retreat Function, the Feed Rate during Rapid Feed

0: Clamped at the equivalent of over10% of the magnification

1: Clamped at over 100% equivalent to magnification

#1 MC4 In the MPG back Function, in the Execution of the G Code Related to the Measurement

#2 MC6 In the MPG back Function, in the Execution of the G Code Related to the Measurement

Set M code accommodation and the number of M codes for each group

00: 3 × 4 groups

10: 4×3 groups

01: 6×2 groups

#6 MGO In the MPG back Function, in the Execution of the G Code Involved in the Measurement

0: MPG pulse is valid

1: MPG pulse is invalid, which is always executed at no more than 100% override speed

	#7	#6	#5	#4	#3	#2	#1	#0
6401	STO							CRH

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 CRH Whether to Prohibit MPG Return in the Manual Control MPG Return Mode

0: Yes

1: No

#7 STO The Timing of S-code and T-code Output when Moving in the Reverse Direction during Manual Hand Pulse Retreat and when Moving in the Forward Direction

0: Different

1: Same

6405

Clamped MPG Retreat Function Fast Moving Speed Multiplier (MLF)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 10

「Valid mode」: Immediate

「Value range」: 0~100

Set the override value used to clamp the rapid traverse speed of the MPG return function. When 0 is set, the function is invalid. The setting of RPO (No.6400#0) is valid (0~100).

6410

The Amount Of Movement Of The Hand Pulse Generator Per Pulse (MPM)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 10

「Valid mode」: Immediate

「Value range」: 0~100

Set the amount of movement (0-100) of the manual pulse generator per pulse by conversion of the magnification.

The amount of mechanical movement when the hand pulse generator is actually rotated can be calculated as follows:

$[\text{Command speed}] \times [\text{MPG override}] \times ([\text{parameter setting value}]/100) \times 8/60000 \text{ (mm or inch)}$

For example: when the command speed is 30mm/min, the MPG override is 100, and the parameter (No.6410) is 1, the amount of movement caused by each pulse of the MPG is calculated according to the following formula.

$[\text{Movement amount per 1 pulse}] = 30[\text{mm/min}] \times 100 \times (1/100) \times (8/60000)[\text{min}] = 0.004\text{mm}$

6411

The M Code Of Group A In Mpg Rollback (AM1)

6412	The M Code Of Group A In Mpg Rollback (AM2)
6413	The M Code Of Group B In Mpg Rollback (BM1)
6414	The M Code Of Group B In Mpg Rollback (BM2)
6415	The M Code Of Group C In Mpg Rollback (CM1)
6416	The M Code Of Group C In Mpg Rollback (CM2)
6417	The M Code Of Group D In Mpg Rollback (DM1)
6418	The M Code Of Group D In Mpg Rollback (DM2)
6419	The M Code Of Group E In Mpg Rollback (EM1)
6420	The M Code Of Group E In Mpg Rollback (EM2)
6421	The M Code Of Group F In Mpg Rollback (FM1)
6422	The M Code Of Group F In Mpg Rollback (FM2)

「Parameter type」：Font type

「Modify authority」：Equipment

「Parameter default」：0

「Valid mode」：Immediate

「Value range」：0~100

This parameter sets the M code of the output group when moving in the reverse direction.

1.25 Parameters related to graphic display

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

「Parameter type」：Bit type

「Modify authority」：Equipment

「Parameter default」：00000000

「Valid mode」：Immediate

#3 DPA Graphic Display Screen, the Display of the Current Position

0: Display the actual position considering tool compensation and offset

1: Display the programmed position without considering tool compensation and offset

1.26 Parameters related to display of running time and number of parts

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

「Modify authority」: Equipment

「Factory default」: 0000 0000

#0 PCMM Code to Count the Total Number of Processed Parts and the Number of Processed Parts

0: M code specified by M02, M30 and parameter NO.6710

1: Only use the M code specified by parameter NO.6710

#1 PRT Signal PRTSF (F62.7) Indicating the Number of Processed Parts Reached when Resetting

0: Turn off

1: No shutdown

#1 PRA When the Processed Part Count Reaches

0: No alarm generated

1: An alarm generated

6710	M Code To Count The Total Number Of Processed Parts And The Number Of Processed Parts (MPC)
-------------	--

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~9999

When the processing program executes the M code set by this parameter, the total number of processed parts and the number of processed parts increase by 1.

Note: When the set value is 0, it is invalid (M00 cannot count the number of parts). Also cannot set 98, 99 and 198.

6713

Number Of Parts Required (RPM)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~99999999

Set the number of parts (0~99999999) required for the number of output parts to reach the signal PRTSF (F62.7).

Note: When the set value is 0, the number of parts is regarded as infinite, and PRTSF is not output.

1.27 Parameters related to tool life management

	#7	#6	#5	#4	#3	#2	#1	#0
6800			GRC	GRS	SIG	LTM	GS2	GS1

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Power on

#0 GS1 According to the Maximum Number of Groups Set in Parameter 6813, the Combination of the Number of Groups that can be Registered through the Settings of this Parameter GS1 and GS2 and the Number of Tools per Group can be Changed

#1 GS2 According to the Maximum Number of Groups Set in Parameter 6813, the Combination of the Number of Groups that can be Registered through the Settings of GS1 and GS2 and the Number of Tools per Group can be Changed

The relationship between the parameters GS1 and GS2 and the number of tools is shown in the following table.

GS2	GS1	Number of Groups	Number of Tools
0	0	1~1/8 of the maximum number of groups (No. 6813)	1~16
0	1	1~1/4 of the maximum number of groups (No. 6813)	1~8
1	0	1/2 of 1~ the maximum number of groups (No. 6813)	1~4
1	1	1~the maximum number of groups (No.6813)	1~2

#2 LTM Specification of Tool Life Count Type

0: Specify according to the number of times

1: Specify according to time

#3 SIG In the Signal-based Tool Skip

0: Execute skip in the currently executing group

1: Specified by external tool group number selection signal

#4 GRS When Inputting Tool Change Reset Signal TLRST

0: The clear group is specified by GRC of parameter 6800#5

1: Clear the execution data of all registered groups

#5 GRC When the Tool Change Reset Signal TLRST is Input, the Specified Group

0: The group whose life is exhausted is automatically detected by CNC

1: Specified by the external tool group number selection signal

	#7	#6	#5	#4	#3	#2	#1	#0
6801		TCT				LVF	TSM	

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#1 TSM In the Tool Life Management Function, There is a Life Count in the Case of Multiple Offset Commands

0: Count according to the same tool number

1: Count according to each tool

#2 When LVF Uses Time to Count the Life Value in the Tool Life Management Function, Set the Tool Life Counting Magnification Signal *TLVO ~ *TLV9<G049.0~G050.1>

0: Invalid

1: Valid

#6 TCT In the Batch Wear Compensation Function, Tool Wear Compensation is in Accordance with

0: Compensation for exhaustion of life

1: Compensation every time

	#7	#6	#5	#4	#3	#2	#1	#0
6802	RMT							T99

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 T99 When M99 in the Main Program is Executed, when there is a Tool Group that has Expired

0: Do not output tool change signal

1: Output the tool change signal and enter the automatic operation stop state

#7 RMT Tool Life Forecast Signal TLCHB

0: The remaining amount of life (life value-life counter), $\leq$ when the redesign value is ON, the remaining amount of life $>$ the redesign value is OFF

1: Remaining amount of life = ON when redesigned value, OFF when remaining amount of life $\neq$ redesigned value

	#7	#6	#5	#4	#3	#2	#1	#0
6804		LFI						

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#6 LFI In Tool Life Management, the Life Count of the Selected Tool

0: Valid

1: Through the tool life counting invalid signal LFCIV <G48.2>, the switch is valid and invalid

	#7	#6	#5	#4	#3	#2	#1	#0
6805	TSLP	TSLW	TSLM	TSLA			FGL	

「Factory default」: 0000 0000

「Modify authority」: Equipment

「Valid mode」: Immediate

#1 FGL When the Life Count Type is the Specified Time, in the Life Data Registration Based on G10

0: In units of 1 minute

1: In units of 0.1 seconds

#4 TSLA Simple tool life alarm output

0: Immediate

1: End

#5 TSLM Simple Tool Life M Code Output

0: Off

1: On

#6 TSLW When the Simple Tool Life is Reached

0: Alarm

1: Prompt

#7 TSLP Simple Tool Life Forecasting Function

0: Off

1: On

6810**The Ignored Number For Tool Life Management (TLC)**

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~99999999

When a value exceeds the value set, instructed by a T code here, the value obtained by subtracting some set numbers from the value of the T code becomes the tool group number for the management of tool life-span .

6811**M Code For Restarting Tool Life Count (MRN)**

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~127

If the life value is set by the number of passes, the tool group whose life has expired when the tool life count is commanded and the M code is restarted. Even if there is one, the tool change signal (TLCH) will be output. When it is set to 0, this parameter will be ignored.

6813**Maximum Number of Groups for Tool Life Management (MTN)**

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Power on

「Value range」: 0, 8, 16, 32, 64, 128

Set the maximum number of groups used in each path. After setting this parameter, you need to

temporarily cut off the power.

6820

Tool Life Management

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~3

Set the tool life management mode: 0: not used, 1: normal mode, 2: simple mode, 3: automatic wear compensation mode.

6844

The Remaining Life Of The Tool (TLP)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~65535

When the tool life is specified, the remaining tool life (number of times of use) of the output tool life reaching forecast signal is output.

6845

Tool Remaining Life Time (TLR)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~4300

When the tool life is specified, the remaining tool life (usage time) of the output tool life reaching forecast signal is output.

6850

Simple Tool Life M Codes

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~9999

Set the M code for simple tool life counting. A setting of 0 indicates M31.

1.28 Parameters on position switch

6910	Execute The Control Axis Of The 1 st Position Switch (PSA1)
6911	Execute The Control Axis Of The 2 nd Position Switch (PSA2)
6912	Execute The Control Axis Of The 3 rd Position Switch (PSA3)
6913	Execute The Control Axis Of The 4 th Position Switch (PSA4)
6914	Execute The Control Axis Of The 5 th Position Switch (PSA5)
6915	Execute The Control Axis Of The 6 th Position Switch (PSA6)
6916	Execute The Control Axis Of The 7 th Position Switch (PSA7)
6917	Execute The Control Axis Of The 8 th Position Switch (PSA8)
6918	Execute The Control Axis Of The 9 th Position Switch (PSA9)
6919	Execute The Control Axis Of The 10 th Position Switch (PSA10)
6920	Execute The Control Axis Of The 11 th Position Switch (PSA11)
6921	Execute The Control Axis Of The 12 th Position Switch (PSA12)
6922	Execute the control axis of the 13 th position switch (PSA13)
6923	Execute The Control Axis Of The 14 th Position Switch (PSA14)
6924	Execute The Control Axis Of The 15 th Position Switch (PSA15)
6925	Execute The Control Axis Of The 16 th Position Switch (PSA16)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~total control axis number

Set the control axis number of the corresponding position switch (0~total control axis number).

When the setting value is zero, it means that the position switch function is not used.

6930	The Maximum Value Of The Operating Range Of The 1 st Position Switch (PAMAX1)
6931	The Maximum Value Of The Operating Range Of The 2 nd Position Switch (PAMAX2)
6932	The Maximum Value Of The Operating Range Of The 3 rd Position Switch (PAMAX3)
6933	The Maximum Value Of The Operating Range Of The 4 th Position Switch (PAMAX4)
6934	The Maximum Value Of The Operating Range Of The 5 th Position Switch (PAMAX5)
6935	The Maximum Value Of The Operating Range Of The 6 th Position Switch (PAMAX6)
6936	The Maximum Value Of The Operating Range Of The 7 th Position Switch (PAMAX7)
6937	The Maximum Value Of The Operating Range Of The 8 th Position Switch (PAMAX8)
6938	The Maximum Value Of The Operating Range Of The 9 th Position Switch (PAMAX9)
6939	The Maximum Value Of The Operating Range Of The 10 th Position Switch (PAMAX10)
6940	The Maximum Value Of The Operating Range Of The 11 th Position Switch (PAMAX11)
6941	The Maximum Value Of The Operating Range Of The 12 th Position Switch (PAMAX12)

6942	The Maximum Value Of The Operating Range Of The 13 th Position Switch (PAMAX13)
6943	The Maximum Value Of The Operating Range Of The 14 th Position Switch (PAMAX14)
6944	The Maximum Value Of The Operating Range Of The 15 th Position Switch (PAMAX15)
6945	The Maximum Value Of The Operating Range Of The 16 th Position Switch (PAMAX16)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Setting Unit	Data Unit	Valid Range	
		IS-B	IS-C
Metric machine	mm	-999999.999~999999.999	-99999.9999~99999.9999
Imperial machine	inch	-99999.9999~99999.9999	-9999.99999~9999.99999

Set the maximum value of the corresponding position switch action range (-99999.9999~99999.9999).

6950	The Minimum Value Of The Operating Range Of The 1 st Position Switch (PAMIN1)
6951	The Minimum Value Of The Operating Range Of The 2 nd Position Switch (PAMIN2)
6952	The Minimum Value Of The Operating Range Of The 3 rd Position Switch (PAMIN3)
6953	The Minimum Value Of The Operating Range Of The 4 th Position Switch (PAMIN4)

6954	The Minimum Value Of The Operating Range Of The 5 th Position Switch (PAMIN5)
6955	The Minimum Value Of The Operating Range Of The 6 th Position Switch (PAMIN6)
6956	The Minimum Value Of The Operating Range Of The 7 th Position Switch (PAMIN7)
6957	The Minimum Value Of The Operating Range Of The 8 th Position Switch (PAMIN8)
6958	The Minimum Value Of The Operating Range Of The 9 th Position Switch (PAMIN9)
6959	The Minimum Value Of The Operating Range Of The 10 th Position Switch (PAMIN10)
6960	The Minimum Value Of The Operating Range Of The 11 th Position Switch (PAMIN11)
6961	The Minimum Value Of The Operating Range Of The 12 th Position Switch (PAMIN12)
6962	The Minimum Value Of The Operating Range Of The 13 th Position Switch (PAMIN13)
6963	The Minimum Value Of The Operating Range Of The 14 th Position Switch (PAMIN14)
6964	The Minimum Value Of The Operating Range Of The 15 th Position Switch (PAMIN15)
6965	The Minimum Value Of The Operating Range Of The 16 th Position Switch (PAMIN16)

「Modify authority」 : Equipment

「Valid mode」 : Immediate

「Parameter type」 : Font type

「Factory default」 : 0

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch

Set the minimum value of the corresponding position switch action range (-99999.9999~99999.9999).

1.29 Parameters related to MPG feed

	#7	#6	#5	#4	#3	#2	#1	#0
7001					DMI			MIT

「Parameter type」 : Bit type

「Modification authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

#0 MIT Manual intervention and return function

0: Invalid

1: Valid

#3 DMI Cancel manual intervention and return data in edit mode

0: Invalid

1: Valid

7010	Speed For Manual Intervention And Return (TRFR)
-------------	--

「Parameter type」 : Font type-real number type

「Modification authority」 : Equipment

「Parameter default」 : 5000.000

「Valid mode」 : Immediate

「Value range」 : 6.0000~60000.0000

Set the speed for manual intervention and return. Unit: 1mm/min for metric machine tools; 1inch/min for imperial machine tools.

7011	FI Speed For Manual Intervention And Return (TRFRL)
-------------	--

「Parameter type」 : Font type-real number type

「Modification authority」: Equipment

「Parameter default」: 1000.000

「Valid mode」: Immediate

「Value range」: 6.0000~60000.0000

Set the FL speed for manual intervention and return. Unit: 1mm/min for metric machine tools; 1inch/min for imperial machine tools.

	#7	#6	#5	#4	#3	#2	#1	#0
7100			MPX		HCL			JHD

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 JHD In Manual Mode, MPG Feed or Incremental Feed in MPG Feed Mode

0: Invalid

1: Valid

	When JHD=0		When JHD=1	
	Manual mode	MPG mode	Manual mode	MPG mode
Manual feed	O	×	O	×
MPG feed	×	O	O	O
Incremental feed	×	×	×	O

#3 HCL Enable the Display of MPG Interruption Amount to be Cleared by Soft Key Operation

0: Invalid

1: Valid

#5 MPX MPX Feed Rate Selection Signal

0: All MPGs use the same magnification signal MP1, MP2

1: Each MPG uses independent multiplying signal, the first MP1 uses MP1 and MP2, and the second MPG uses MP21 and MP22.

	#7	#6	#5	#4	#3	#2	#1	#0
7102								HNGx

「Parameter type」: Bit axis type

「Modification authority」: Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 HNGx The Movement Direction of Each Axis and the Rotation Direction of the Hand Pulse Generator

0: Same

1: Opposite

	#7	#6	#5	#4	#3	#2	#1	#0
7103						HNT	RTH	

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#1 RTH Whether to cancel the handwheel feed interruption amount via reset or emergency stop.

0: Do not cancel

1: Cancel

#2 HNT Incremental Feed/MPG Feed Moving Amount Override is Set to the Override Selected in the MPG Feed Moving Amount Selection Signal

0: 1 Times

1: 10 Times

7110	Is Handwheel Pulse Generator 1 Valid (EMP)
-------------	---

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Immediate

Set whether handwheel pulse generator 1 is valid.

0: Invalid

1: Valid

7111	Is Handwheel Pulse Generator 2 Valid (EMP2)
-------------	--

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

Set whether handwheel pulse generator 2 is valid.

0: Invalid

1: Valid

7113

MPG Feed Rate M (MFM)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 1~1000

Set the override when MP1=0 and MP2=1 for MPG feed movement selection signal.

7114

MPG Feed Rate N (MFN)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 1000

「Valid mode」: Immediate

「Value range」: 1~1000

Set the handwheel feed rate when the handwheel feed movement selection signals are MP1=1 and MP2=1.

Movement Amount Selection Signal		Movement Amount (hand pulse feed)
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 Amount During Mpg Feed (APM)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~999999

When the handwheel feed exceeds the maximum handwheel movement speed instantaneously, the pulses that exceed the rapid movement are not canceled but stored instead.

This parameter sets the allowable value for this storage amount.

Note: When a larger magnification such as $\times 100$ is selected and the handwheel pulse generator is rotated rapidly, causing the handwheel feed to exceed the maximum feed speed of the handwheel, the speed will be limited to the maximum feed speed of the handwheel. The pulses that exceed the maximum feed speed of the handwheel are ignored, resulting in a discrepancy between the scale value of the handwheel pulse generator and the actual movement amount.

In this case, if an allowable value is set in this parameter in advance, the pulses that exceed the rapid traverse speed will not be canceled but will be temporarily stored in the CNC (however, the part exceeding the allowable value will be ignored). When the rotation speed of the handwheel slows down or stops, the stored pulses will be output as movement commands.

Special attention should be paid that if the allowable value is set too large, even if the handwheel has stopped rotating, the CNC will continue to move until all the stored pulses are executed before stopping.

7131**MPG Feed Rate M2 (MFM2)**

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 1~1000

Set the override for MP21 = 0 and MP22=1 when the MP21 is selected for the feed movement amount selection signal.

7132**MPG Feed Rate N2 (MFN2)**

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 1000

「Valid mode」: Immediate

「Value range」: 1~1000

Set the multiplying factor when MP21=1 and MP22=1 for the selection signal of MP21 feed movement amount.

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

7300**MOU****MOA**

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#6 MOA During program restart, before moving to the reprocessing start position

0: Output the last MSTB code

1: Output all M codes and the last S code

#7 MOU During program restart, after searching for the program to be restarted and before moving to the reprocessing start position

0: Do not output MSTB code

1: Output MSTB code

	#7	#6	#5	#4	#3	#2	#1	#0
7301								ROF

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 ROF In the restart coordinate display of the program restart screen, set whether to display various tool compensation values

0: Display with tool offset compensation considered

1: Depends on the settings of parameters DAL (No.3104#6) and DAC (No.3104#7)

7310	Axis Sequence For Rapid Traverse Movement During Program Restart (ROAX)							
-------------	--	--	--	--	--	--	--	--

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 1

「Valid mode」 : Immediate

After starting the program restart, specify the order of axes (1 to number of controlled axes) that will move to the restart point via rapid traverse.

1.30 Parameters related to polygon processing

	#7	#6	#5	#4	#3	#2	#1	#0
7603	PSS	PQS		PSM				PPCE

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 PPCE Phase compensation function validity in polygon machining

0: Invalid

1: Valid

#4 PSM Operating mode of the workpiece rotation axis in polygon machining

0: Speed mode

1: Position mode

#6 PQS The PQ value in polygon machining is

0: Rotational speed ratio between the tool rotation axis and the workpiece rotation axis

1: Ratio of the number of polygons to the number of tools

#7 PSS Acceleration/deceleration time constant of the workpiece axis during polygon machining

0: Use the tool axis time constant

1: Use the workpiece axis time constant

7610

Control Axis Number Of The Tool Rotation Axis Used For Polygon Machining (PCA)

「Parameter type」 : Font type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~control axis number

Set the control axis number of the tool rotation axis used in the polygon house. When it is set to 0, it means that the function is not enabled.

7621

The Upper Limit Speed Of The Tool Rotation Axis For Polygon Machining (PSM)

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 3000

「Valid mode」 : Immediate

「Value range」 : 0~9999

Set the upper limit rotation speed of the tool rotation axis.

7623

**Set The Acceleration And Deceleration Time Constant Of The Tool Axis
In Polygon Machining (PAT)**

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 0~4000

Set the acceleration/deceleration time constant of the tool axis (0~4000).

7624

**Set The Acceleration And Deceleration Time Constant Of The Workpiece
Axis In Polygon Machining (SPAT)**

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 0~4000

Set the acceleration/deceleration time constant of the workpiece axis (0~4000).

7625

**Set The Phase Check Completion Width Of The Tool Axis In Polygon
Machining (PTRE)**

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

Set the complete width of the phase check of the tool axis in polygon machining (0~99.9999).

When it is set to 0, it means no check.

Valid Range	
IS-B	0~99.999
IS-C	0~99.9999

7626

**Set The Phase Check Completion Width Of The Workpiece Axis In
Polygon Processing (PWRE)**

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

Set the phase check completed width of the workpiece axis in polygon processing (0~99.9999).

When it is set to 0, it means no check.

Valid Range	
IS-B	0~99.999
IS-C	0~99.9999

1.31 Parameters about electronic gearbox

	#7	#6	#5	#4	#3	#2	#1	#0
7700						HDR		

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#2 HDR Helical Gear Compensation Compensation Direction

0: Specify a negative value for P

1: Specify a positive value for P

	#7	#6	#5	#4	#3	#2	#1	#0
7703	EPSS							EPCE

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 EPCE Phase Compensation Function in Electronic Gearbox

0: Invalid

1: Valid

#7 EPSS The Acceleration and Deceleration Time Constant of the Workpiece Axis in the Electronic Gear Box

0: Use tool axis time constant

1: Use the workpiece axis time constant

7710	The Axis Number Of The Synchronous Axis When Setting The Electronic Gear Box Function (EAS)
------	---

- 「Parameter type」 : Word axis type
- 「Modification authority」 : Machine tool
- 「Parameter default」 : 0
- 「Valid mode」 : Power on
- 「Value range」 : 0~number of control axes

The main control axis (tool axis) is set to 0, and the parameter value of the slave control axis (tool axis) is set to the main control axis (tool axis) number (0~the number of control axes).

7711

Set The Axis Number Of The Axial Feed Axis In The Helical Gear (ESIA)

- 「Parameter type」 : Word axis type
- 「Modification authority」 : Machine tool
- 「Parameter default」 : 0
- 「Valid mode」 : Power on
- 「Value range」 : 0~number of control axes

When this parameter is set to 0, it means that the Z-axis becomes the axial feed axis (0~the number of control axes).

7721

Set The Upper Limit Speed Of The Tool Axis In The Electronic Gear Box (EPSM)

- 「Parameter type」 : Font type
- 「Modify authority」 : Equipment
- 「Parameter default」 : 3000
- 「Valid mode」 : Immediate
- 「Value range」 : 0~9999

Set the upper limit speed of the tool axis (0~9999).

7723

Set The Acceleration And Deceleration Time Constant Of The Tool Axis In The Electronic Gear Box (ERPAT)

- 「Parameter type」 : Font type
- 「Modify authority」 : Equipment
- 「Parameter default」 : 100
- 「Valid mode」 : Immediate
- 「Value range」 : 0~4000

Set the acceleration/deceleration time constant of the tool axis (0~4000).

7724

Set The Acceleration/Deceleration Time Constant Of The Workpiece Axis In The Electronic Gearbox (ESPAT).

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~4000

Set the acceleration/deceleration time constant of the workpiece axis (0~4000).

7725

Set The Phase Check Completion Width Of The Tool Axis In The Electronic Gear Box (EPTRE)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

Set the complete width of the phase check of the tool axis in the electronic gear box (0~99.9999). When it is set 0 means no check.

Valid Range	
IS-B	0~99.999
IS-C	0~99.9999

7726

Set the phase check completion width of the workpiece axis in the electronic gear box (EPWRE)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 :

Set the phase verification completion width of the workpiece axis in the electronic gear box (0~99.9999). When it is set to 0, it means no verification.

Valid Range	
IS-B	0~99.999
IS-C	0~99.9999

1.32 Parameters related to PLC Axis Control

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000101

「Valid mode」: Immediate

#0 MLE is Valid for the Machine Lock Signal MLK of the PLC Control Axis

0: Valid

1: Invalid

#2 OVE and Signals related to Dry Run and Override used by PLC Axis Control

0: Same as the signal used by CNC

1: Signal specially used for PLC

#3 RDE In PLC Axis Control, Whether Dry Run is Valid for Rapid Traverse Command

0: Invalid

1: Valid

#5 NCC When the PLC Control Axis (the axis selected by the control axis selection signal) is Commanded to Move by the Program Command

0: When the PLC controls the axis according to the axis control command, a P/S (No. 139) alarm occurs; when the axis is not controlled, the CNC command is valid

1: P/S (No.139) alarm

#6 AUX Auxiliary Function Command Code Output Format

0: Single byte

1: Double byte

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000001

「Valid mode」: Immediate

#0 RPD PLC Control Axis's Rapid Movement Speed

0: The feed rate set by parameter No.1420

1: The feed rate specified by the feed rate data in the axis control command

#4, #5 PR1, PR2 In the PLC Axis Control, Set the Unit of Feed Speed per Revolution

PF2	PF1	Speed
0	0	1/1
0	1	1/10
1	0	1/100
1	1	1/1000

#6, #7 FR1, FR2 In the PLC Axis Control, Set the Command Unit of the Feed Speed per Revolution of the Cutting Feed

FR2	FR1	Metric Input (mm)	Imperial Input (inch)
0	0	0.0001mm/rev	0.000001inch/rev
1	1		
0	1	0.001mm/rev	0.00001inch/rev
1	0	0.01mm/rev	0.0001inch/rev

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Immediate

#2 JFM The Unit of the Feed Speed for the Continuous Feed (06h) of the PLC Control Axis

Incremental System	JFM	Metric Input (mm)	Imperial Input (inch)	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 DSL When PLC axis selection is invalid, perform axis selection switching

0: Do not execute and alarm

1: The axis of the uncommanded channel executes the file

#6 NCI In-position Check during PLC Axis Deceleration

0: Proceed

1: Do not proceed

	#7	#6	#5	#4	#3	#2	#1	#0
8005						R10	CDI	

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Immediate

#1 CDI In the PLC Axis Control, When the Diameter of the PLC Axis is Specified

0: The moving distance is specified by the radius

1: The moving distance is specified by diameter programming

#2 R10 Parameter RPD (No.8002#0) is Set to "1", the Command Unit of the Rapid Traverse Speed of the PLC Axis

0:×1

1:×10

	#7	#6	#5	#4	#3	#2	#1	#0
8006	EAL			EFD				

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Immediate

#4 EFD It is the Command Unit of the Feed Rate Data when Cutting Feed (feed per minute) in PLC Axis Control

0:×1

1:×100

#7 EAL PLC Axis Control, Reset Operation by CNC

0: Maintain the PLC control axis

1: Dismiss the alarm of the PLC control axis

8010	Selection Of Di/Do Group For Each Axis Controlled By Plc (EPSA)
------	---

「Parameter type」: Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~4

The DI/DO groups used by each PLC control axis are shown in the following table.

Value	Descriptions
0	The axis is not controlled by PLC
1	Use DI/DO signal of Group A
2	Use DI/DO signal of Group B
3	Use DI/DO signal of Group C
4	Use DI/DO signal of Group D

8022

**The Maximum Feed Speed Per Revolution Of The PLC Controlled Axis
(EPMF)**

「Parameter type」 : Word axis type-real number type

「Modification authority」 : Machine tool

「Parameter default」 : 1000

「Valid mode」 : Immediate

「Input and output」 : Output related

「Value range」 :

Incremental System	Valid data range	
	IS-B	IS-C
Metric machine	6~15000	6~12000
Imperial machine	6~6000	6~6000
Rotary axis	6~15000	6~12000

Set the maximum feed rate per revolution of the PLC control axis.

8028

**Linear Acceleration/Deceleration Time Constant Of PLC Axis Control
Speed Command (EPAT)**

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~3000 ms

Set the linear acceleration/deceleration time constant (0~3000) ms of each PLC control axis for manual feed by speed command.

Note: If set to "0", the system will not perform acceleration/deceleration control.

8030

PLC Control Axis Reference Point Offset (RPS)

「Parameter type」: Word axis type-real number type

「Modification authority」: Machine tool

「Parameter default」: 0

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Incremental System	Data Unit	Valid Data Range	
		IS-B	IS-C
Metric machine	mm	-999999.999~999999.999	-99999.9999~99999.9999
Imperial machine	inch	-99999.9999~99999.9999	-9999.99999~9999.99999
Rotary axis	deg	-999999.999~999999.999	-99999.9999~99999.9999

This parameter sets the reference point offset of the PLC control axis.

1.33 Parameters related to basic functions

8130

Total Control Axis (TCA)

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 2

「Valid mode」: Power on

「Value range」: 1~8

Set the total number of control axes of the CNC system, including NC axis and PLC axis (1~8).

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

「Parameter type」: Bit type

「Modification authority」: Machine tool

「Parameter default」: 00000001

「Valid mode」: Power on

#0 HPG Whether the MPG Feed is Used

0: Not used

1: Used

#7 TEP Whether the Temperature Compensation Function Used

0: Not used

1: Used

	#7	#6	#5	#4	#3	#2	#1	#0
8132						UIC		

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Power on

#2 UIC Whether to Use a Custom Interface

0: Not used

1: Used

	#7	#6	#5	#4	#3	#2	#1	#0
8133	CMW	SMD	FCH					SSC

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 00000001

「Valid mode」 : Power on

#0 SSC Whether to Use the Constant Line Speed Control (G96) Function

0: Not used

1: Used

#5 FCH Whether to Open Quick Tool Change Function

0: Off

1: On

#6 SMD Whether to Open the Machine Tool Diagnosis Function

0: Off

1: On

#7 CMW Whether to Open Custom Program Page

0: Off

1: On

	#7	#6	#5	#4	#3	#2	#1	#0
8135	RPTH					NSQ		NHI

「Parameter type」 : Bit type
 「Modify authority」 : Equipment
 「Parameter default」 : 00000100
 「Valid mode」 : Power on

#0 NHI Whether to Use the MPG Interrupt Function

0: Not used
 1: Used

#2 NSQ Whether to Use the Program Restart

0: Used
 1: Not used

#7 RPTH Whether the Thread Repair Function shall be used

0: Not used
 1: Used

	#7	#6	#5	#4	#3	#2	#1	#0
8136					NOP			

「Parameter type」 : Bit type
 「Modification authority」 : Machine tool
 「Parameter default」 : 00000000
 「Valid mode」 : Power on

#3 NOP Whether to use the soft machine tool panel

0: Not used
 1: Used

8140	Tool Breakage Detection Function (TDG)							
------	--	--	--	--	--	--	--	--

「Parameter type」 : Font type
 「Modification authority」 : Machine tool
 「Parameter default」 : 0
 「Valid mode」 : Power on

This parameter sets the tool breakage detection function mode. 0: Disable, 1: Detect current value, 2: Detect current load.

1.34 Parameters related to inclined axis control

	#7	#6	#5	#4	#3	#2	#1	#0
8200						AZR		AAC

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Power on

#0 AAC Whether to Perform Inclined Axis Control

0: Do not proceed

1: Proceed

#2 AZR When Performing the Manual Reference Point Return Operation of the Inclined Axis in the Inclined Axis Control

0: The orthogonal axis also moves at the same time

1: Orthogonal axis does not move

	#7	#6	#5	#4	#3	#2	#1	#0
8209								ARF

「Parameter type」 : Bit type

「Modify authority」 : Equipment

「Parameter default」 : 00000000

「Valid mode」 : Power on

#0 ARF The Movement from the Intermediate Point of the G28/G30 Command in the Inclined Axis Control to the Reference Point is

0: Action of tilted coordinate system

1: Action in Cartesian coordinate system

8210	Tilt Angle Of Inclined Axis In Inclined Axis Control (INA)
-------------	---

「Parameter type」 : Font type-real number type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

This parameter sets the inclined axis angle in inclined axis control.

Valid Data Range	
IS-B	IS-C
-180.000~180.000	-180.0000~180.0000

8211	The Axis Number Of The Inclined Axis For Inclined Axis Control (ANS)
8212	The Axis Number Of The Orthogonal Axis For Inclined Axis Control (ANC)

- 「Parameter type」 : Font type
- 「Modification authority」 : Machine tool
- 「Parameter default」 : 0
- 「Valid mode」 : Power on
- 「Value range」 : 0~number of control axes

This parameter sets the axis number of the inclined axis when performing inclined axis control. When any parameter is set to 0, or when the same number is set, or when the number of non-control axes is set, it means that the function is invalid.

1.35 Parameters related to synchronous control of feed axis

	#7	#6	#5	#4	#3	#2	#1	#0
8303						SAFx		

- 「Parameter type」 : Bit axis type
- 「Modify authority」 : Equipment
- 「Parameter default」 : 00000000
- 「Valid mode」 : Immediate

#2 SAFx Whether the Movement of the Control Axis is Added to the Timing Speed Display in the Feed Axis Synchronization Control

0: Do not add

1: Add it

	#7	#6	#5	#4	#3	#2	#1	#0
8304			SCAx					

- 「Parameter type」 : Bit axis type
- 「Modification authority」 : Machine tool
- 「Parameter default」 : 00000000
- 「Valid mode」 : Power on

#5 SCAx In the Infeed Axis Synchronization Control

0: Whether to execute synchronous operation is determined by the feed axis synchronization control selection signal of the slave control axis

1: Consistent performance of m synchronous operation

8311	The Axis Number Of The Main Control Axis In The Synchronous Control Of Each Feed Axis (FMT)							
------	--	--	--	--	--	--	--	--

- 「Parameter type」 : Word axis type
- 「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~control axis number

This parameter sets the axis number of the main control axis corresponding to the axis. If the axis is the main control axis, the parameter is set to 0.

8314

**The Maximum Amount Of Error During The Synchronization Error
Check Of Each Feed Axis Is Based On The Machine Coordinate Value
(MESC)**

「Parameter type」 : Word axis type-real number type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	0~99999.999	0~9999.9999	mm
Imperial machine	0~9999.9999	0~999.99999	inch
Rotary axis	0~99999.999	0~9999.9999	deg

This parameter sets the maximum amount of error when the machine coordinate value is used for synchronization error checking. When it is 0, the synchronization error check is not used.

1.36 Parameters related to additional axis

	#7	#6	#5	#4	#3	#2	#1	#0
8400	AOTS	ABFA			AIAL			AMCE

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Immediate

**#0 AMCE When the Additional Axis is Equipped with an Incremental Encoder,
whether the Coordinate System is Memorized when Each Axis is Powered on**

0: Do not memorize

1: Memorize

**#3 AIAL When the Additional Axis Generates an Alarm, the Operating State of
the Main Program**

0: Stop

1: Do not stop

#6 ABFA When a Stored Stroke Detection 1 Alarm Occurs

0: The tool stops after entering the prohibited area

1: The tool stops before entering the prohibited area

#7 AOTS Whether to Output an Overtravel Alarm Signal to the PLC when a Stored Stroke Detection Alarm Occurs

0: No output

1: Output

	#7	#6	#5	#4	#3	#2	#1	#0
8401						ATJE	ACAF	ACTD

「Parameter type」: Bit type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 ACTD Whether reset clears the torque control output value

0: Controlled by G signal

1: Clear

#1 ACAF Whether reset clears the stack full signal

0: Do not clear

1: Clear

#2 ATJE Whether the auxiliary axis torque control multiplier in manual mode is valid

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
8402								AOT2x

「Parameter type」: Bit axis type

「Modify authority」: Equipment

「Parameter default」: 00000000

「Valid mode」: Immediate

#0 AOT2x Stored Stroke Detection 2 Setting of Each Additional Axis

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
8403	ADJZx		AHJZx	AZMIx	ADECx			AHNGx

「Parameter type」 : Bit axis type

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Immediate

#0 AHNGx The Movement Direction of Each Additional Axis and the Rotation Direction of the Manual Pulse Generator

0: Same

1: Opposite

#3 ADECx Each Additional Axis Reference Point Return Deceleration Signal

0: Decelerate when the signal is "0"

1: Decelerate when the signal is "1"

#4 AZMIx Reference Point Return Direction of Each Additional Axis

0: Positive direction

1: Negative direction

#5 AHJZx When the Reference Point of Each Additional Axis is Established and the Reference Point is Manually Returned Again

0: Use deceleration block

1: It has nothing to do with the deceleration block

#7 ADJZx The Setting of Zero Return without Stop in the Control of Each Additional Axis

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
8404							AROSx	AROTx

「Parameter type」 : Bit axis type, additional axis

「Modification authority」 : Machine tool

「Parameter default」 : 00000000

「Valid mode」 : Power on

#0, #1 AROSx AROTx Sets Each Axis Type in the Additional Axis Control

00 linear axis

01 Rotary axis (type A)

11 Rotary axis (type B)

	#7	#6	#5	#4	#3	#2	#1	#0
8407						APOD _x		ARCA _x

「Parameter type」: Bit axis type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#0 ARCA_x Individual auxiliary axis removal function

0: Invalid

1: Valid

#2 APOD_x Set the pulse output direction of each axis in auxiliary axis control

0: Do not invert

1: Invert

	#7	#6	#5	#4	#3	#2	#1	#0
8408			AAPC _x	AAPZ _x				

「Parameter type」: Bit axis type

「Modification authority」: Machine tool

「Parameter default」: 00000000

「Valid mode」: Power on

#4 AAPZ_x When the Absolute Position Detector is in the Additional Axis Control, the Position of the Mechanical Position and the Absolute Position Detector

0: Inconsistent

1: Consistent

#5 AAPC_x Set the Position Detector of Each Axis in the Additional Axis Control

0: Do not use absolute encoder

1: Use absolute encoder

8410	Axis Number Setting For Additional Axis Control (ASS)
------	---

「Parameter type」: Font type

「Modification authority」: Machine tool

「Parameter default」: 0

「Valid mode」: Power on

「Value range」: 0~4

This parameter sets the total number of axes in the additional axis control. When this parameter

is set to 0, it means that the additional axis function is not enabled.

8411**Load Program Number For Additional Axis Control (APL)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~9999

This parameter sets the program number loaded during additional axis control.

8412**Programming Name Of Additional Axis Control (ASPnx)**

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 88

「Valid mode」 : Immediate

「Value range」 : 88, 89, 90, 65, 66, 67, 85, 86, 87

Set the axis name of each additional axis (X=88, Y=89, Z=90, A=65, B=66, C=67, U=85, V=86, W=87).

8413**Additional Axis Controls The Attributes Of Each Axis In The Basic Coordinate System (ANSAx)**

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~3

Set the attributes of each additional axis in the basic coordinate system.

8414**Station Address For Additional Axis Control (AASAx)**

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~99

Set the station address of each additional axis.

8417

Command Multiplication Ratio For Additional Axis Control (Numerator Of Gear Ratio) (ACMRx)

「Parameter type」: Word axis type

「Modification authority」: Machine tool

「Parameter default」: 1

「Valid mode」: Power on

「Value range」: 1~65536

The command multiplication ratio of each additional axis (the numerator of the gear ratio).

8418

Detection Multiplication Ratio Of Additional Axis Control (Gear Ratio Denominator) (ADMRx)

「Parameter type」: Word axis type

「Modification authority」: Machine tool

「Parameter default」: 1

「Valid mode」: Power on

「Value range」: 1~65536

Set the detection multiplication ratio (gear ratio denominator) of each additional axis.

8419

As The Additional Axis Is Equipped With An Absolute Encoder, The Allowable Deviation Value Of The Machine Coordinate System Is Established When The Machine Is Powered On (AMERx)

「Parameter type」: Word axis type-real number type

「Modification authority」: Machine tool

「Parameter default」: 0.1

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	0~9.999	0~9.9999	mm
Imperial machine	0~0.9999	0~0.99999	inch
Rotary axis	0~9.999	0~9.9999	deg

This parameter sets the additional axis to be used for deviation alarm detection when the machine tool coordinate system is set up after power-on. When it is set to 0, the deviation will not be detected.

8420

Whether Additional Axis Torque Control Is Valid (ANCTx)

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Valid mode」 : Power on

「Value range」 : 0~1

Set whether the torque control of each additional axis is valid (0~1; 0 means invalid, 1 means valid).

8421

**The Detection Time Of Arrival Signal For Additional Axis In G35
Command (ATCTx)**

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 200

「Valid mode」 : Immediate

「Value range」 : 200~2000

Set the detection time (200~2000) ms of the arrival signal for each additional axis commanded by G35.

8422

**Additional Axis Torque Controls F Command Speed Command
Numerator (ANCMx)**

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Immediate

「Value range」 : 1~99999

The speed command numerator (1~99999) of each additional axis torque set to control F command.

8423

**Speed Command Denominator Of Additional Axis Torque Control F
Command (ANDMx)**

「Parameter type」 : Word axis type

「Modification authority」 : Machine tool

「Parameter default」 : 1

「Valid mode」 : Immediate

「Value range」 : 1~99999

Set the speed command denominator (1~99999) of each additional axis torque control F command.

8424

**Command Speed Of Torque Control Of Each Additional Axis In Manual
Mode (AHNCSx)**

「Parameter type」 : Word axis type

「Modification authority」: Machine tool

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 6~6000

Set the command speed (6~6000) for torque control of each additional axis.

8425

The Command Torque Of Each Additional Axis Torque Control In Manual Mode (AHNCNx)

「Parameter type」: Word axis type

「Modification authority」: Machine tool

「Parameter default」: 400

「Valid mode」: Immediate

「Value range」: 10~30000

Set the command torque for torque control of each auxiliary axis (10~30000) $\times 0.01\text{N} \cdot \text{m}$.

8426

Additional Axis Backlash Compensation Amount (ABCVx)

「Parameter type」: Word axis type-real number type

「Modification authority」: Machine tool

「Parameter default」: 0

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	-9.999~9.999	-9.9999~9.9999	mm
Imperial machine	-0.9999~0.9999	-0.99999~0.99999	inch

Set the backlash compensation amount of each additional axis.

8427

Additional Axis Backlash Compensation Frequency Value (ACPFx)

「Parameter type」: Word axis type

「Modification authority」: Machine tool

「Parameter default」: 32

「Valid mode」: Immediate

「Value range」: 1~128

Set the backlash compensation frequency value of each additional axis.

8430**Additional Axis Waits For The Minimum Value Of The M Code Range
(AWMI)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0, 100~999

Set the minimum value of the waiting M code range between the additional axis and the main program. When the setting value is 0, it is invalid.

8431**The Maximum Range Of The Additional Axis Waiting For The M Code
(AWMM)**

「Parameter type」 : Font type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0, 100~999

Set the maximum value of the waiting M code range between the additional axis and the main program. When the setting value is 0, it is invalid.

8434**Each Additional Axis Is Assigned The X Address Of The Deceleration
Signal Used For The Reference Point Return Operation (AXBx)**

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」 : 0~127

Set the X address of the deceleration signal allocated to each additional axis for reference point return operation.

8435**The Bit Address Of The X Address Of The Deceleration Signal For Each
Additional Axis Used For The Reference Point Return Operation (AXTx)**

「Parameter type」 : Font axis type additional axis

「Modify authority」 : Equipment

「Parameter default」 : 0

「Valid mode」 : Immediate

「Value range」: 0~7

Set the bit address of the X address of the deceleration signal allocated to each additional axis for reference point return operation.

8440

**The Coordinate Value Of The Positive Boundary Of The Additional Axis
Stored Stroke Detection (APCx)**

「Parameter type」: Word axis type-real number type

「Modification authority」: Machine tool

「Parameter default」: 0

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch
Rotary axis	-999999.999~999999.999	-99999.9999~99999.9999	deg

Set the coordinate value of the positive direction of each additional axis stored stroke detection at the boundary of the machine coordinate system.

8441

**The Coordinate Value Of The Negative Direction Boundary Of The
Additional Axis Stored Stroke Detection (ANCx)**

「Parameter type」: Word axis type-real number type

「Modification authority」: Machine tool

「Parameter default」: 0

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch
Rotary axis	-999999.999~999999.999	-99999.9999~99999.9999	deg

Set the boundary coordinate value in the machine tool coordinate system for the negative direction of the stored stroke detection of each additional axis.

8442

**The Coordinate Value Of The Positive Boundary Of The Additional Axis
Stored Stroke Detection 2 (APC2x)**

「Parameter type」 : Word axis type-real number type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch

Set the boundary coordinate value of each additional axis stored stroke detection 2 in the machine coordinate system in the positive direction.

8443

**Additional Axis Stored Stroke Detection 2 Negative Direction Boundary
Coordinate Value (ANC2x)**

「Parameter type」 : Word axis type-real number type

「Modification authority」 : Machine tool

「Parameter default」 : 0

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch
Rotary axis	-999999.999~999999.999	-99999.9999~99999.9999	deg

Set the boundary coordinate value of each additional axis stored stroke detection 2 in the machine tool coordinate system in the negative direction.

8450

Rapid Traverse Speed Of Each Additional Axis (ARTx)

「Parameter type」 : Word axis type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 8000

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	30~100000	30~60000	mm
Imperial machine	30~48000	30~24000	inch
Rotary axis	30~100000	30~60000	deg

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

8451

The Minimum Speed Of Rapid Movement Of Each Additional Axis (AF0x)

「Parameter type」: Word axis type-Real number type Additional axis

「Modify authority」: Equipment

「Parameter default」: 400

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	30~100000	30~60000	mm
Imperial machine	30~48000	30~24000	inch
Rotary axis	30~100000	30~60000	deg

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

8452

Manual Feed Speed Of Each Additional Axis (AJFx)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 1000

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	30~100000	30~60000	mm
Imperial machine	30~48000	30~24000	inch
Rotary axis	30~100000	30~60000	deg

Set the feed speed of each additional axis during manual continuous feed (manual feed).

8453

The Maximum Speed Of Mpg Feed For Each Additional Axis (AHMx)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 400

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	30~100000	30~60000	mm
Imperial machine	30~48000	30~24000	inch
Rotary axis	30~100000	30~60000	deg

Set the maximum feed rate of MPG feed for each additional axis.

8454

Auxiliary Axis Manual Rapid Feed Rate (AMRRx)

「Parameter type」: Font type-real number type

「Modify authority」: Equipment

「Parameter default」: 2000.0000

「Valid mode」: Immediate

「Value range」: 30.0000~60000.0000

Set the feed rate for manual rapid continuous feed (manual feed) of each auxiliary axis.

8456

Reference Point Return Speed Of Each Additional Axis (ARPx)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 4000

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	30~100000	30~60000	mm
Imperial machine	30~48000	30~24000	inch
Rotary axis	30~100000	30~60000	deg

Set the reference point return speed of each additional axis.

8457

Reference Point Return Fl Speed Of Each Additional Axis (AFLx)

「Parameter type」: Word axis type-real number type

「Modify authority」: Equipment

「Parameter default」: 400

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	30~100000	30~60000	mm
Imperial machine	30~48000	30~24000	inch
Rotary axis	30~100000	30~60000	deg

Set the speed (FL speed) of each additional axis after deceleration when returning to the reference point.

8459

Maximum Cutting Feed Speed Of Additional Axis (AFR)

「Parameter type」: Font type-real number type

「Modification authority」: Machine tool

「Parameter default」: 8000

「Input and output」: Output related

「Valid mode」: Immediate

「Value range」:

Value Range	IS-B	IS-C	Unit
Metric machine	6~100000	6~60000	mm
Imperial machine	6~48000	6~24000	inch

Set the maximum cutting feed speed of the additional axis, the unit: 1mm/min for metric machine tools; 0.1inch/min for inch machine tools.

8460

Acceleration And Deceleration Time Constant For Rapid Traverse Of Each Additional Axis (ATTx)

「Parameter type」: Word axis type

「Modify authority」: Equipment

「Parameter default」: 100

「Valid mode」: Immediate

「Value range」: 0~4000

Set the acceleration and deceleration time constant of each additional axis rapid traverse.

8462

Acceleration And Deceleration Time Constant Of Manual Feed For Each Additional Axis (AJEx)

「Parameter type」 : Word axis type

「Modify authority」 : Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~4000

Set the acceleration/deceleration time constant for manual feed of each additional axis.

8463

Acceleration And Deceleration Time Constant Of Mpg Feed For Each Additional Axis (AMEx)

「Parameter type」 : Font axis type additional axis

「Modify authority」 : Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~4000

Set the acceleration and deceleration time constant of the MPG feed of each additional axis.

8464

The Acceleration/Deceleration Time Constant Of The Cutting Feed Of Each Additional Axis (AATx)

「Parameter type」 : Font axis type additional axis

「Modify authority」 : Equipment

「Parameter default」 : 100

「Valid mode」 : Immediate

「Value range」 : 0~4000

Set the acceleration/deceleration time constant of the cutting feed of each additional axis.

8468

In-Position Width For Auxiliary Axis Rapid Feed (APWRx)

「Parameter type」 : Font type-real number type

「Modification authority」 : Equipment

「Parameter default」 : 5.0000

「Valid mode」 : Immediate

「Value range」 : 0~9999.9999

Set the in-position width for rapid feed of each auxiliary axis (0~999.9999). No in-position detection will be performed when the set value is 0.

8469

In-Position Width For Auxiliary Axis Cutting Feed (APWFx)

「Parameter type」 : Font type-real number type

「Modification authority」: Equipment

「Parameter default」: 5.0000

「Valid mode」: Immediate

「Value range」: 0~9999.9999

Set the in-position width for cutting feed of each auxiliary axis (0~999.9999). No in-position detection will be performed when the set value is 0.

8470	Additional Control Axis That Performs The 1st Position Switch Function (APSW1)
8471	Additional Control Axis That Performs The 2nd Position Switch Function (APSW2)
8472	Additional Control Axis That Performs The 3rd Position Switch Function (APSW3)
8473	Additional Control Axis That Performs The 4th Position Switch Function (APSW4)
8474	Additional Control Axis That Performs The 5th Position Switch Function (APSW5)
8475	Additional Control Axis That Performs The 6th Position Switch Function (APSW6)
8476	Additional Control Axis That Performs The 7th Position Switch Function (APSW7)
8477	Additional Control Axis That Performs The 8th Position Switch Function (APSW8)

「Parameter type」: Font type

「Modify authority」: Equipment

「Parameter default」: 0

「Valid mode」: Immediate

「Value range」: 0~Maximum number of additional axis control axes

Set the additional control axis number for executing each position switch. When the set value is 0, it

means invalid.

8478	The Maximum Value Of The Operating Range Of The 1 st Position Switch (APSM1)
8479	The Maximum Value Of The Operating Range Of The 2 nd Position Switch (APSM2)
8480	The Maximum Value Of The Operating Range Of The 3 rd Position Switch (APSM3)
8481	The Maximum Value Of The Operating Range Of The 4 th Position Switch (APSM4)
8482	The Maximum Value Of The Operating Range Of The 5 th Position Switch (APSM5)
8483	The Maximum Value Of The Operating Range Of The 6 th Position Switch (APSM6)
8484	The Maximum Value Of The Operating Range Of The 7 th Position Switch (APSM7)
8485	The Maximum Value Of The Operating Range Of The 8 th Position Switch (APSM8)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch

Set the maximum value of the operating range of each position switch.

8486	The Minimum Value Of The Operating Range Of The 1 st Position Switch (APSI1)
8487	The Minimum Value Of The Operating Range Of The 2 nd Position Switch (APSI2)
8488	The Minimum Value Of The Operating Range Of The 3 rd Position Switch (APSI3)
8489	The Minimum Value Of The Operating Range Of The 4 th Position Switch (APSI4)
8490	The Minimum Value Of The Operating Range Of The 5 th Position Switch (APSI5)
8491	The Minimum Value Of The Operating Range Of The 6 th Position Switch (APSI6)
8492	The Minimum Value Of The Operating Range Of The 7 th Position Switch (APSI7)
8493	The Minimum Value Of The Operating Range Of The 8 th Position Switch (APSI8)

「Parameter type」 : Font type-real number type

「Modify authority」 : Equipment

「Parameter default」 : 0

「Input and output」 : Output related

「Valid mode」 : Immediate

「Value range」 :

Value Range	IS-B	IS-C	Unit
Metric machine	-999999.999~999999.999	-99999.9999~99999.9999	mm
Imperial machine	-99999.9999~99999.9999	-9999.99999~9999.99999	inch

Set the minimum value of the operating range of each position switch.

1.37 Parameters related to GSK-Link communication function

	#7	#6	#5	#4	#3	#2	#1	#0
9000	GPL							GNET

「Parameter type」 : Bit type

「Modification authority」 : Machine tool

「Parameter default」 : 00000001

「Valid mode」 : Power on

#0 GNET System GSK-Link Communication Function is Valid

0: Invalid

1: Valid

#7 GPL Data length of system GSK-Link cyclic communication

0: 8 bytes

1: 12 bytes

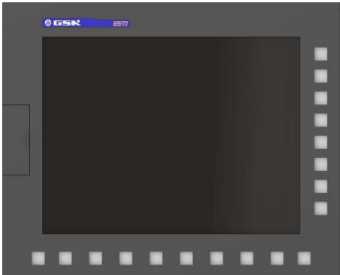
Appendix II Installation Layout

2.1 Button Structure (10.4-inch)

Serial Number	Order Model	Appearance	Dimensions (Width × Height)	Remarks
1	GSK25Ti-V10CMC (CNC Controller)		290mm × 340mm	10.4-inch vertical aluminum alloy panel; Crystal keys with color zoning; Host with vertical soft function keys
2	GSK25Ti-MPU-V10BRMC (Operation Panel Unit)		290mm × 280mm	Equipped with 10.4-inch vertical host; Crystal keys with color zoning; Operation panel without handwheel.
3	GSK25Ti-H10CDC (CNC Controller)		400mm × 220mm	10.4-inch horizontal aluminum alloy panel; Crystal keys with color zoning; Host with vertical soft function keys.
4	GSK25Ti-MPU-H10ARDC (Operation Panel Unit)		400mm × 250mm	Equipped with 10.4-inch horizontal host; Crystal keys with color zoning; Operation panel with handwheel.

Serial Number	Order Model	Appearance	Dimensions (Width × Height)	Remarks
5	GSK25Ti-MPU-H10BRDC (Operation Panel Unit)		400mm × 250mm	Equipped with 10.4-inch horizontal host; Crystal keys with color zoning; Operation panel without handwheel.

2.2 Button Structure (15-inch)

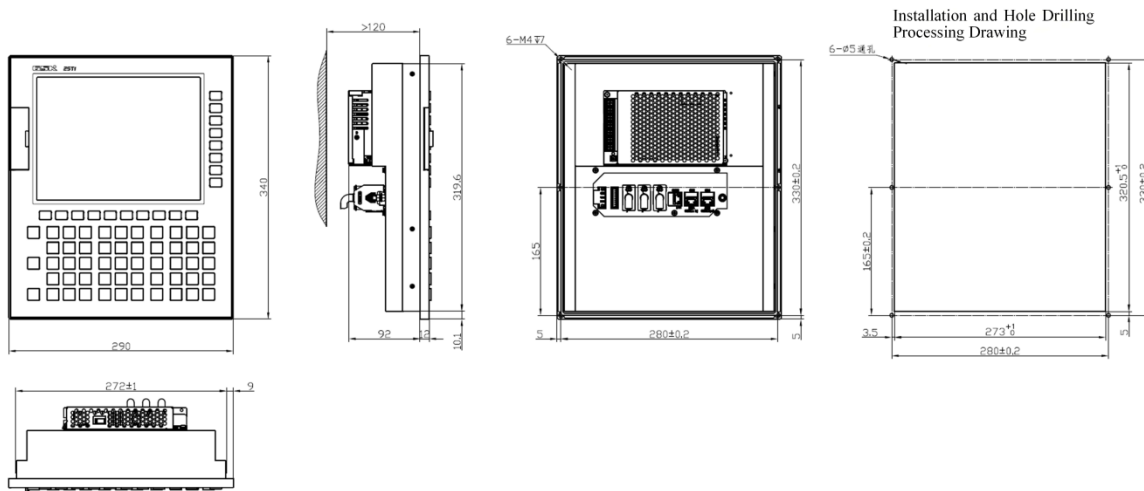
Serial Number	Order Model	Appearance	Dimensions (Width × Height)	Remarks
1	GSK25Ti-V15CDC (CNC Controller)		400mm × 320mm	15-inch vertical aluminum alloy panel; Crystal keys with color zoning; Host with vertical soft function keys
2	GSK25Ti-MPU-V15CDC (Operation Panel Unit)		400mm × 320mm	Equipped with 15-inch vertical host; Crystal keys with color zoning; Operation panel.

2.3 I/O Unit Models

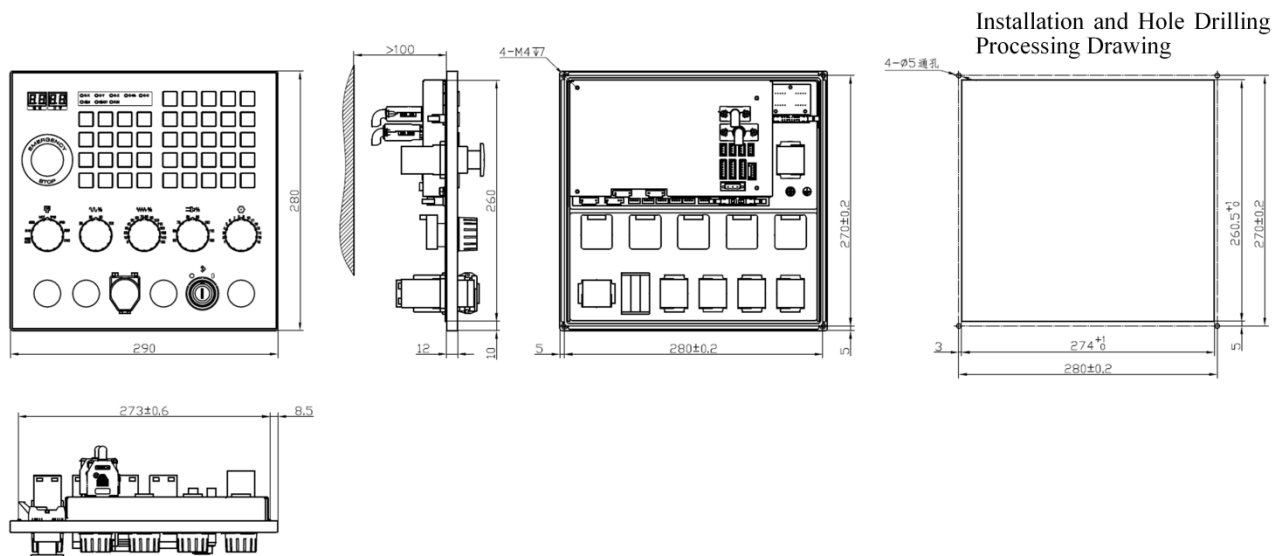
Serial Number	Order Model	Appearance	Dimensions (Width × Height)	Remarks
1	IOR-44F		63mm × 270mm × 148mm	● Header socket interface type, output is high level; ● 48-point input / 32-point output; ● 4 channels of 0~10V analog voltage output; ● Optional splitter MCT07.
2	IOR-04T		90.2mm × 305mm × 80mm	● Terminal interface type, output is low level; ● 48-point input / 32-point output; ● No analog voltage output interface.
3	IOR-44T		90.2mm × 305mm × 80mm	● Terminal interface type, output is low level; ● 4 channels of 0~10V analog voltage; ● 48-point input / 32-point output.
4	MCT07			Equipped with IOR-44F.

2.4 Vertical Button Structure Installation Dimension Diagram (10.4-inch)

● Controller Appearance and Installation Dimension Diagram

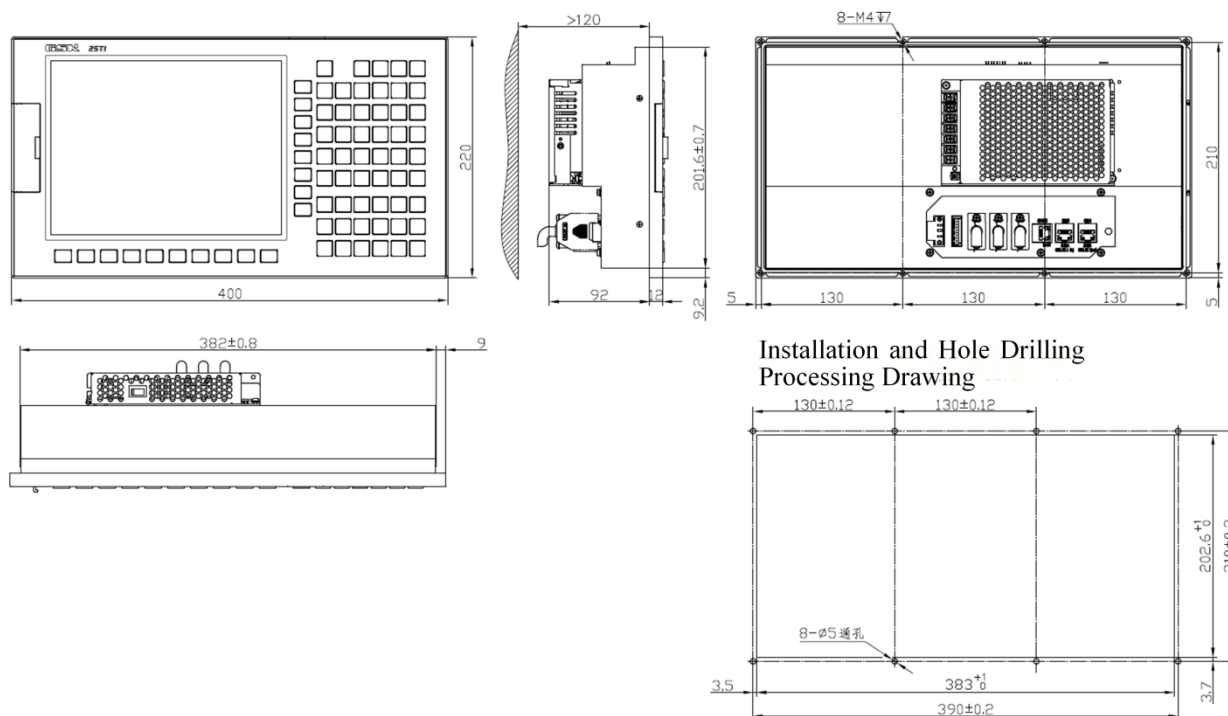


● Operation Panel Appearance and Installation Dimension Diagram

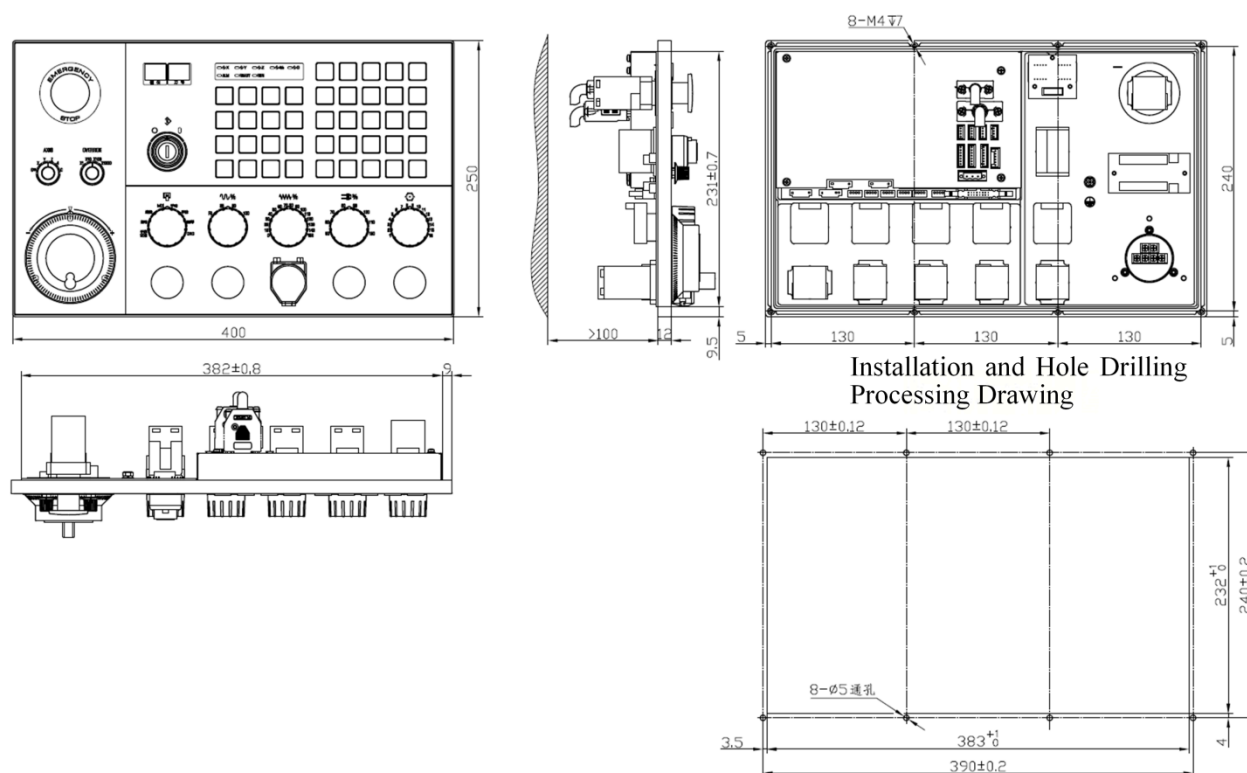


2.5 Horizontal Button Structure Installation Dimension Diagram (10.4-inch)

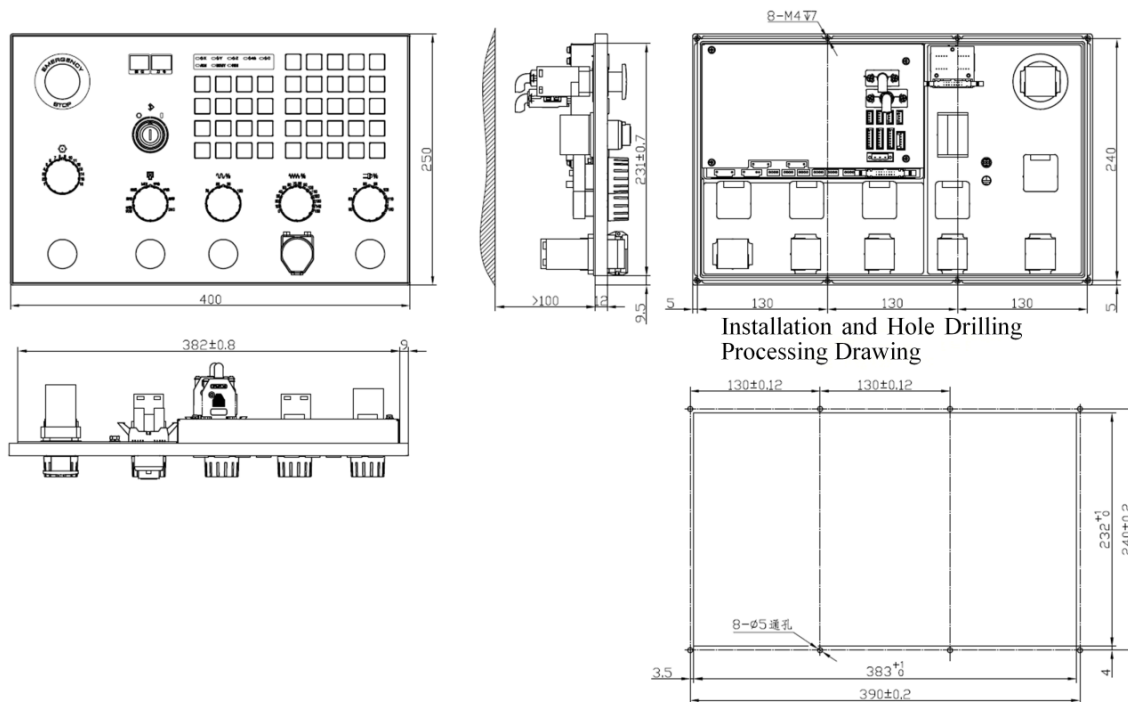
● Controller Appearance and Installation Dimension Diagram



● Operation Panel Appearance and Installation Dimension Diagram (with handwheel)

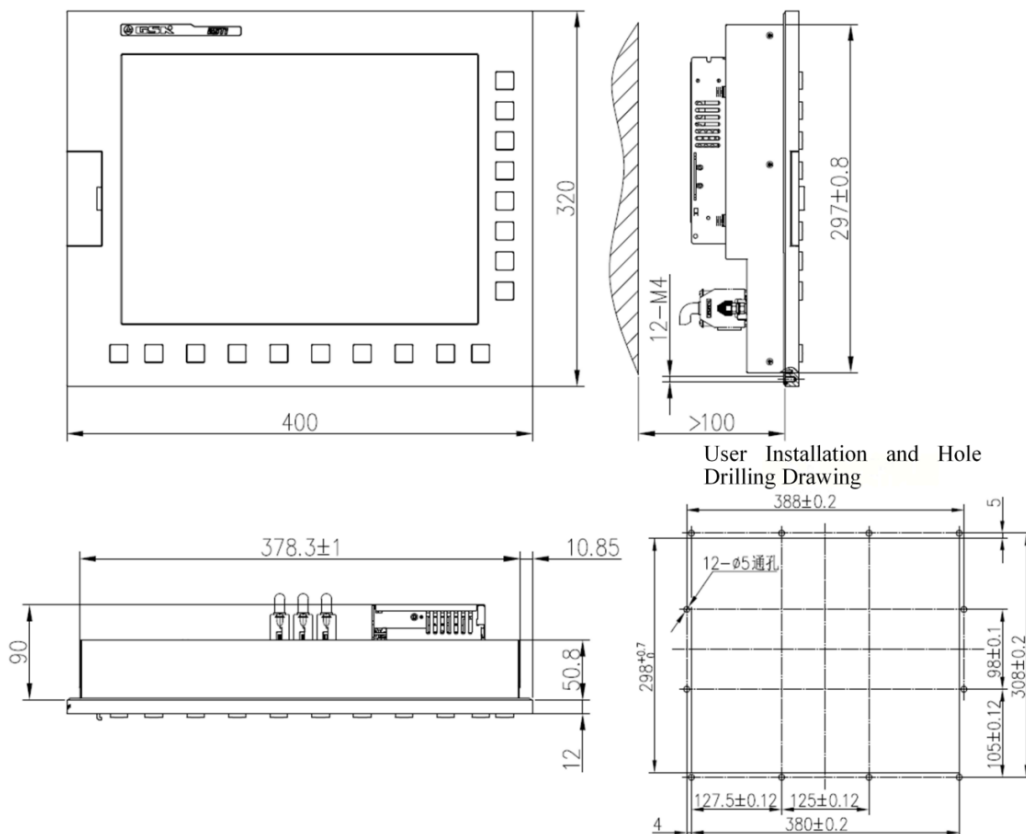


● Operation Panel Appearance and Installation Dimension Diagram (without handwheel)

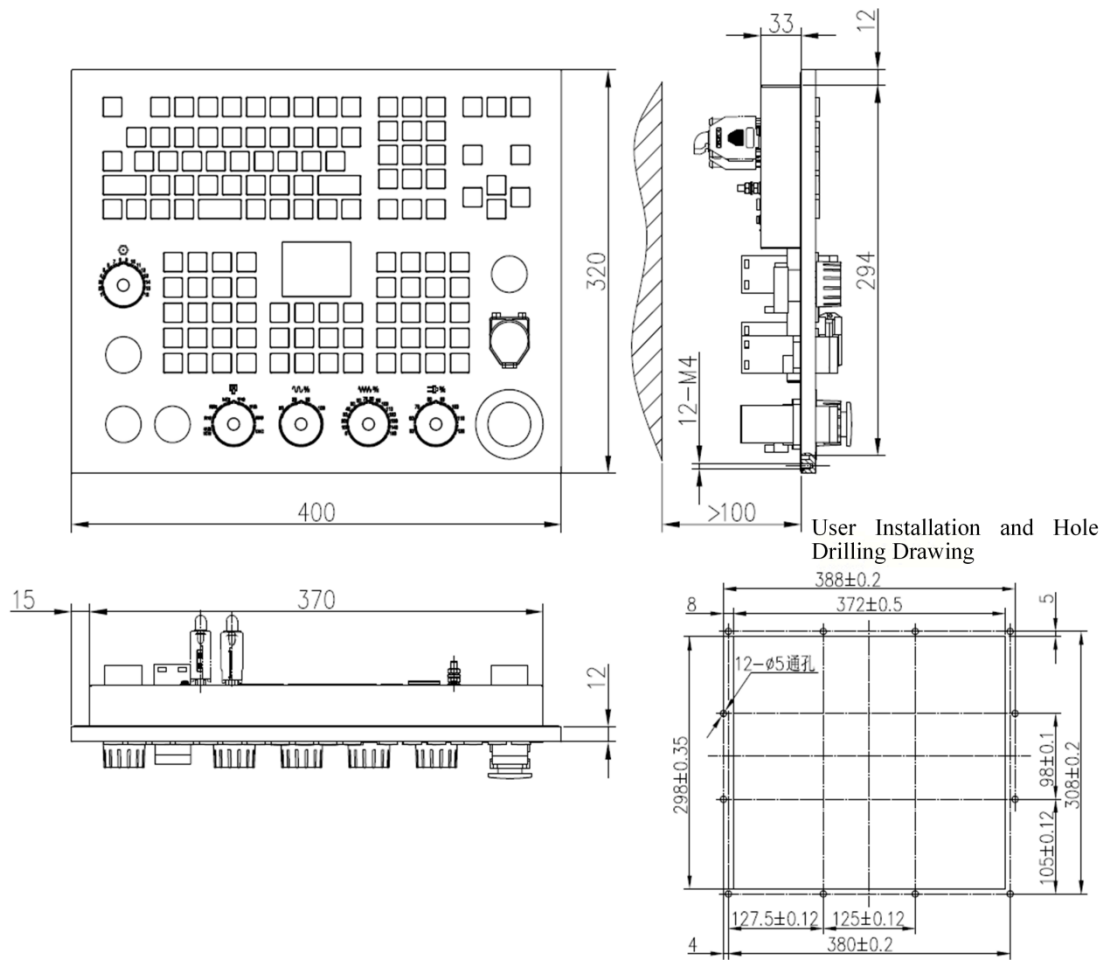


2.6 Vertical Button Structure Installation Dimension Diagram (15-inch)

● Controller Appearance and Installation Dimension Diagram

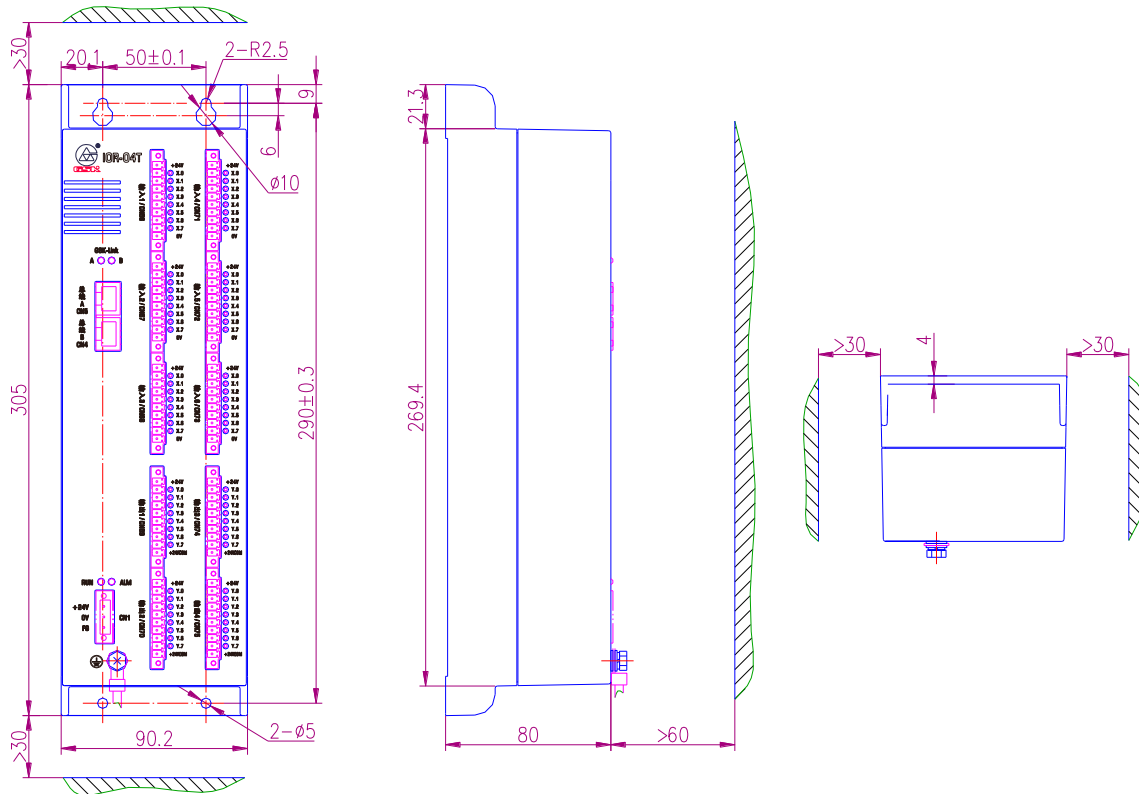


- Operation Panel Appearance and Installation Dimension Diagram (without handwheel)

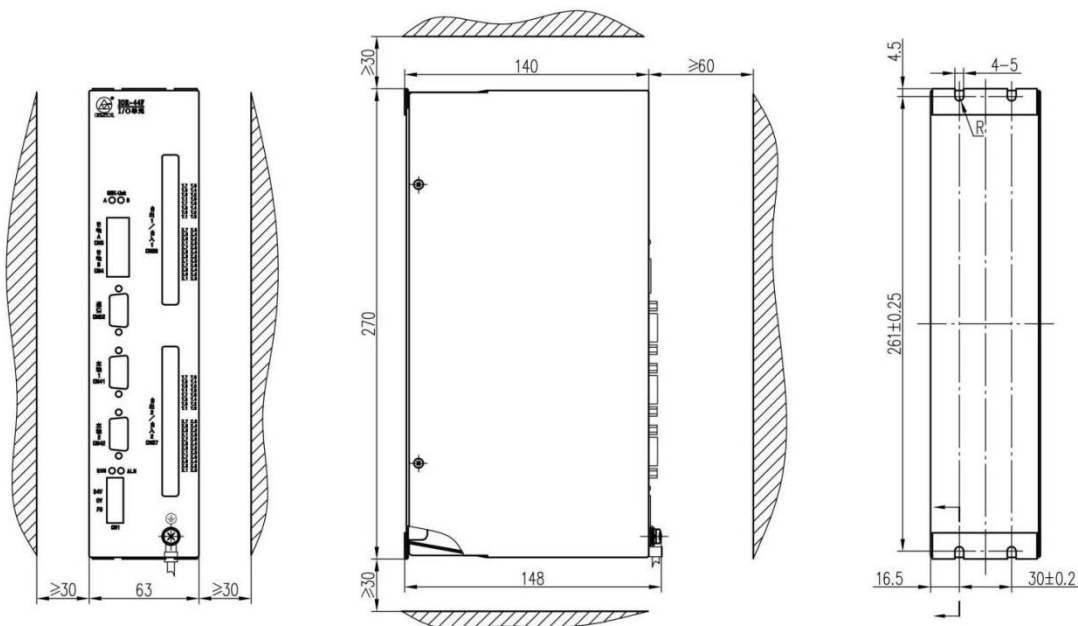


2.7 I/O Unit Overall Dimensions

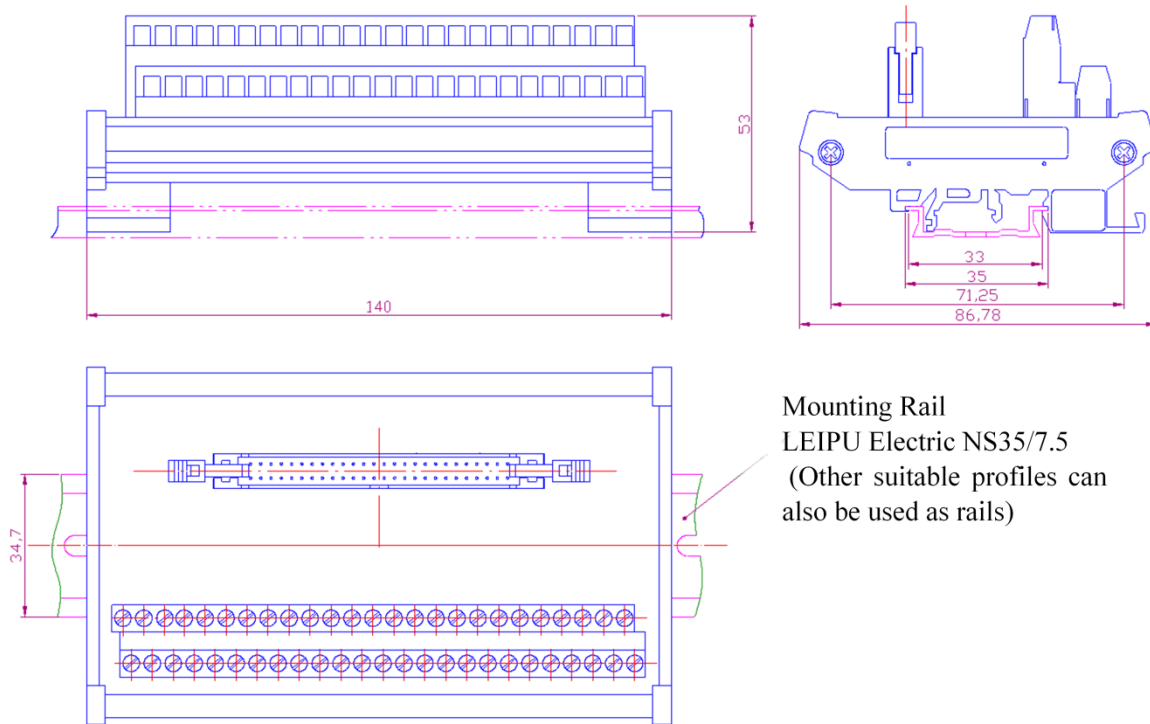
2.7.1 Installation Method and Overall Dimensions of IOR-04T/IOR-44T



2.7.2 Installation Method and Overall Dimensions of IOR-44F



2.7.3 Installation Method and Overall Dimensions of MCT07 (Compatible with IOR-21F/IOR-44F)



Appendix III Standard Ladder Diagram Function Configuration

3.1 The definition of the key address of the machine tool panel

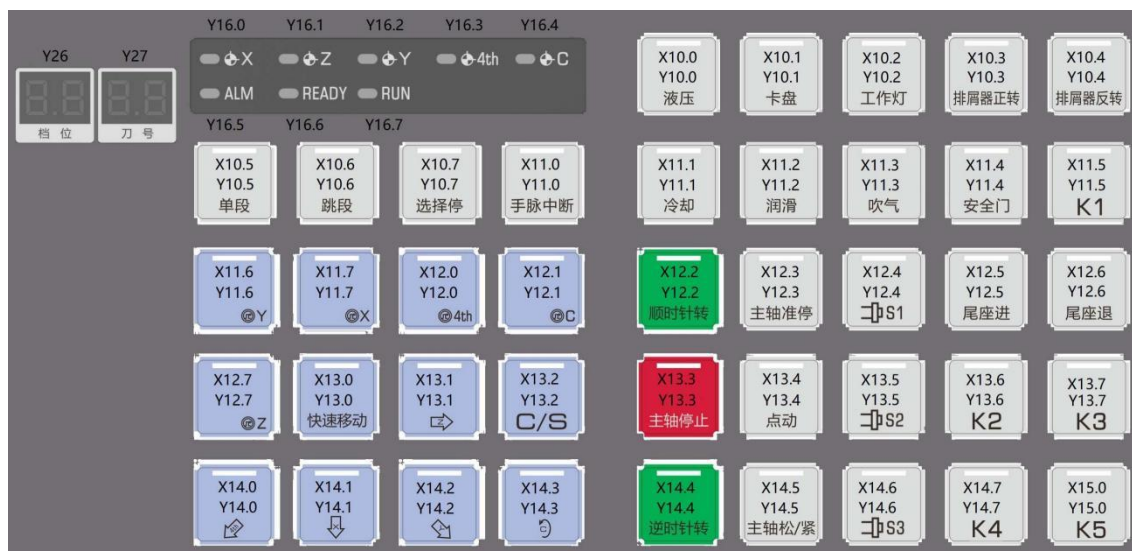


Fig. 3-1 Addresses of Independent-Key Machine Tool Panel (10.4-inch)



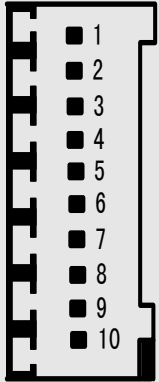
Fig. 3-2 Addresses of Independent-Key Machine Tool Panel (15-inch)

3.2 The definition of standard ladder diagram X and Y address

The I/O of the system is divided into high-speed I/O signals and general-purpose I/O signals. The high-speed I/O signals are those of the CN61 interface on the CNC rear cover. The general-purpose I/O signals are those on the extended I/O units. The functions of the CNC's I/O signals (except for those marked with fixed addresses) are defined by the system's built-in PLC program (ladder diagram). When the system is installed on a machine tool, the I/O functions are determined by the design of the machine tool manufacturer. For details, please refer to the instruction manual of the machine tool manufacturer.

It should be noted that in this section, the functions of general-purpose I/O signals (i.e., X and Y addresses) are described for the standard PLC program of the system!

3.2.1 High-speed I/O interface

 CN61 pin	Corresponding Connector Pin	PLC Address	Function Defined by Standard PLC Address		Note
	CN61.1		+24V		
	CN61.2	X0.0		Spare	
	CN61.3	X0.1		Spare	
	CN61.4	X0.2		Spare	
	CN61.5	X0.3		Spare	
	CN61.6	X0.4	SKIP	G31 jump signal	Fixed address
	CN61.7	X0.5	ESP	Emergency halt of input signal	Fixed address
	CN61.8	X0.6	G36	G36 jump signal	Fixed address
	CN61.9	X0.7	G37	G37 jump signal	Fixed address
	CN61.10		0V		

3.2.2 General Machine Tool I/O Interface

The general-purpose input and output of the system are allocated through the GSK-Link communication interface to I/O units. A maximum of 4 I/O units can be connected via the GSK-Link bus, with each I/O unit having a maximum of 48 input points and 32 output points. The address range is X80~X127 and Y80~Y127, and users need to configure the system addresses according to their requirements.

In the standard configuration of the system, an I/O unit with 48 input points and 32 output points is provided. The configured addresses in the standard ladder diagram of the system are X100~X105 and Y100~Y103. All I/O signal addresses introduced in this manual are based on this standard.

If there are any differences, please refer to the machine tool manual provided by the machine tool manufacturer.

PLC Address	Function Defined by Standard PLC Address		Note
X100.0	SAGT	Protective Door Fully Closed Signal	
X100.1		Protective Door Fully Open Signal	
X100.2	DIQP	Chuck Input Signal	
X100.3		Spare	
X100.4	DITW	Tailstock Control Signal	
X100.5		Spare	
X100.6	PRES	Pressure Detection Signal	
X100.7		Liuxin Turret Signal E	
X101.0		Turret Released in Position (Liuxin Turret Signal F)	
X101.1		Turret Locked in Position	
X101.2			
X101.3		Low Lubricating Oil Level	
X101.4		Spare	
X101.5	M41I	Gear Shift to 1st Gear in Position	
X101.6	M42I	Gear Shift to 2nd Gear in Position	
X101.7	T01	Tool Position Signal 1 (Liuxin Turret Signal A)	
X102.0	T02	Tool Position Signal 2 (Liuxin Turret Signal B)	
X102.1	T03	Tool Position Signal 3 (Liuxin Turret Signal C)	
X102.2	T04	Tool Position Signal 4 (Liuxin Turret Signal D)	
X102.3	T05	Turret Signal 5	
X102.4			
X102.5		Spare	
X102.6			
X102.7			
X103.0	LMI1+	Axis 1 Positive Overtravel Signal	
X103.1	LMI2+	Axis 2 Positive Overtravel Signal	
X103.2	LMI3+	Axis 3 Positive Overtravel Signal	

PLC Address	Function Defined by Standard PLC Address		Note
X103.3	WQPJ	Chuck In-position Signal (External Clamping/Internal Unclamping In-position)	
X103.4	NQPJ	Chuck In-position Signal (External Unclamping/Internal Clamping In-position)	
X103.5		Spare	
X103.6		Spare	
X103.7		Spare	
X104.0	LMI1-	1 st Axis Negative Overtravel Signal	
X104.1	LMI2-	2 nd Axis Negative Overtravel Signal	
X104.2	LMI3-	3 rd Axis Negative Overtravel Signal	
X104.3	LMI4+	4 th Axis Positive Overtravel Signal	
X104.4	LMI4-	4 th Axis Negative Overtravel Signal	
X104.5	LMI5+	5 th Axis Positive Overtravel Signal	
X104.6	LMI5-	5 th Axis Negative Overtravel Signal	
X104.7		Spare	
X105.0		Spare	
X105.1		Inverter Spindle Speed Reached Signal	
X105.2		Spare	
X105.3		Inverter Spindle Alarm Signal	
X105.4		Spare	
X105.5		Spare	
X105.6		Spare	
X105.7		Spare	
Y100.0	M08/M09	Cooling Output Signal	
Y100.1	M32/M33	Lubrication Output Signal	
Y100.2		Hydraulic Station Output Signal	
Y100.3	M03	Spindle Counterclockwise Rotation Signal (Forward Rotation)	
Y100.4	M04	Spindle Clockwise Rotation Signal (Reverse Rotation)	

PLC Address	Function Defined by Standard PLC Address		Note
Y100.5		Spare	
Y100.6	M35/M36	Spindle Brake Output Signal	
Y100.7		Spare	
Y101.0	M41	Spindle Gear 1 Output Signal	
Y101.1	M42	Spindle Gear 2 Output Signal	
Y101.2	M43	Spindle Gear 3 Output Signal	
Y101.3	M44	Spindle Gear 4 Output Signal	
Y101.4	M12 (DOQPJ)	External Chuck Clamping Output Signal / Internal Chuck Unclamping Output	
Y101.5	M13 (DOQPS)	External Chuck Unclamping Output Signal / Internal Chuck Clamping Output	
Y101.6	TL+	Turret Forward Rotation Output Signal / Delta Mode MD0	
Y101.7	TL-	Turret Reverse Rotation Output Signal / Delta Mode MD1	
Y102.0		Turret Release Output	
Y102.1		Turret Lock Output	
Y102.2	YLAMP	Three-Color Light - Yellow (Normal state, neither running nor alarming)	
Y102.3	GLAMP	Three-Color Light - Green (Running state)	
Y102.4	RLAMP	Three-Color Light - Red (Alarm state)	
Y102.5	M10	Tailstock Forward Output Signal	
Y102.6	M11	Tailstock Retract Output Signal	
Y102.7		Lighting Lamp	
Y103.0	M37	Chip Conveyor Forward Rotation	
Y103.1	M38	Chip Conveyor Reverse Rotation	
Y103.2	M22	Automatic Door Close	
Y103.3	M23	Automatic Door Open	
Y103.4	IDX0	Delta Tool Number Output 0	
Y103.5	IDX1	Delta Tool Number Output 1	

PLC Address	Function Defined by Standard PLC Address		Note
Y103.6	IDX2	Delta Tool Number Output 2	
Y103.7	IDX3	Delta Tool Number Output 3	

3.2.3 Handheld Box Interface

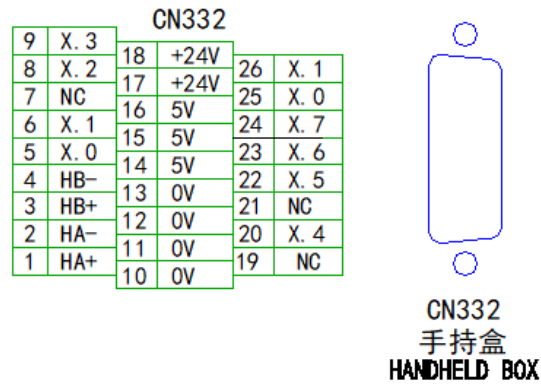


Fig. 3-3 CN332 MPG interface (26-pin D-pin socket)

Corresponding to the DB Header Pin	Signal Definition	Signal Description	Function Defined by Standard PLC Address
CN332.1,CN332.2	HA+, HA-	Handwheel Phase A Signal Input	/
CN332.3,CN332.4	HB+, HB-	Handwheel Phase B Signal Input	/
CN332.5	X37.0	PLC Signal Address, Digital Input	External Handwheel Box X-Axis Selection Signal
CN332.6	X37.1	PLC Signal Address, Digital Input	External Handwheel Box Y-Axis Selection Signal
CN332.7, CN332.19 CN332.21	NC		
CN332.8	X37.2		
CN332.9	X37.3	PLC Signal Address, Digital Input	External Handwheel Box Z-Axis Selection Signal
CN332.22	X37.5	PLC Signal Address, Digital Input	External Handwheel Box ×1 Gear Signal
CN332.23	X37.6	PLC Signal Address, Digital Input	External Handwheel Box ×10 Gear Signal
CN332.24	X37.7	PLC Signal Address, Digital Input	External Handwheel Box ×100 Gear Signal
CN332.25	X38.0	PLC Signal Address, Digital Input	External Handwheel Box ×1000 Gear Signal

Corresponding to the DB Header Pin	Signal Definition	Signal Description	Function Defined by Standard PLC Address
CN332.26	X38.1	PLC Signal Address, Digital Input	External Handwheel Box 4th Axis Selection Signal
CN332.20	X37.4	PLC Signal Address, Digital Input	External Handwheel Box C-Axis Selection Signal
CN332.10, CN332.11 CN332.12, CN332.13	0V	PLC Signal Address, Digital Input	External Handwheel Box 6th Axis Selection Signal
CN332.14, CN332.15 CN332.16	+5V	0V	/
CN332.17, CN332.18	+24V	+5V	/

Note: X37.0~X38.1 are high-level input valid, that is, when the input signal is connected to +24V, the input is valid, and the X address status is 1, otherwise the X address status is 0.

Appendix IV Alarm Handling

4.1 CNC Alarm Handling

Alarm No.	Significance	Possible Causes	Handling Method
000	Emergency stop alarm, ESP input open circuit	ESP input open circuit	Restore the ESP emergency stop signal input to clear the alarm.
001	Failed to open part program	Failed to open part program	Press the reset button to eliminate the alarm and reselect the execution program.
002	Failed to read program line	Failed to read the program line code when MDI was running the program or grammar check.	Press the reset key to clear the alarm.
009	There is a format error with the operator in the program.	The format of the operation instruction in the program segment is incorrect.	Please modify the program
010	There are too many characters in a program.	A program exceeds 256 characters	Please modify the program.
011	The data exceeds the allowable value range	The data input exceeds the allowable value range or the specified data exceeds 8 digits.	Data needs to be modified.
012	Address not found	There is no address at the beginning of the block, only a value or symbol is input.	The program needs to be modified.
013	No data after the address	There is no data immediately after the address or the format of the expression after the address is wrong, and parentheses are not used.	The program needs to be modified.
014	Incorrect use of the minus sign	The symbol "-" is used incorrectly (used where the minus sign cannot be used, or input too many minus signs).	The program needs to be modified.
015	Incorrect use of decimal point	The decimal point "." is used incorrectly (use it after a command that cannot use the decimal point, or input too many decimal points).	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
016	Enter an illegal address	An unusable address was entered in the effective information area.	The program needs to be modified.
017	Incorrect G code	A G code without this function was instructed.	The program needs to be modified.
018	Duplicate address error	If the same address is commanded two or more times in a block or two or more G codes of the same group are commanded in a block, please check parameter No.3403#6 AD2.	The program needs to be modified.
019	The program does not specify an end statement.	The program does not specify an end statement such as M30 or M02.	Modify the program or parameter 3404#6
020	The axis address of the instruction is incorrect.	The axis address specified in the program does not exist. Check parameters No.1020 and No.1025.	The program needs to be modified.
040	Too many M codes appeared	In the same block, the number of commands exceeds the number set by parameter 3404#7 M3B, or exceeds the maximum number of 3.	The program or parameters need to be modified.
041	M command value is out of range	The number of digits of the M command exceeds the setting value of parameter 3030 MCB, or the value of the M code command exceeds the range.	The program or parameters need to be modified.
042	A G code that cannot be shared with M99 is instructed	G28, G30, G53 or G36/G37 cannot be specified in the same block as M99.	The program needs to be modified.
043	The value of P instruction in M or T block is out of range.	The P value in M or T block can only be specified as 1~4.	The program needs to be modified.
050	The commanded tool number is not within the allowable range	A tool number that does not exist is specified, or it exceeds the number of digits set by parameter 3032 TCB.	The program or parameters need to be modified.
051	The compensation number of the command does not exist	The tool offset number selected by T code is out of range (0~99).	The program needs to be modified.
052	No T code is allowed in this block	G10, G04 and G50 of G code of A set or G92 of B set of G code cannot be specified in the same block as T code, please check parameter 5006#1 TGC.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
060	The feedrate setting value is not within the range	The feedrate is not commanded or the feedrate is improper in the cutting feed: 1. When using A set of codes, the F values required for G98 and G99 modals are different, please check G98 and G99 modals; 2. When using the B set of codes, the F values required for G94 and G95 modals are different, please check G94 and G95 modals.	The program or parameters need to be modified.
061	The spindle speed setting value is not within the range	The number of digits of the commanded spindle speed S exceeds the setting value of parameter No.3031 SCB, or the commanded value exceeds the range.	The program or parameters need to be modified.
062	Cannot use G96 command	When parameter No.8130#0 (whether to use the constant line speed control G96 function) is set to not use the constant line speed control, G96 is commanded.	The program or parameters need to be modified.
063	Axis command error in spindle constant surface speed control	In G96 mode, the calculation reference axis set by parameter No.3770 does not exist. If parameter No.3770 is 0, check whether any control axis in parameter No.1022 is set to 0 (X-axis of the basic coordinate system).	Parameter No.1022 or parameter No.3770 needs to be modified.
064	The feed rate setting value in G99 mode is out of range.	In G99 mode, the feed rate is not specified in the cutting feed or the feed rate is out of range (0.001~500mm/r, 0.001~9.99inch/r).	The program needs to be modified.
065	The S value in the spindle speed floating command is incorrect.	In the spindle speed floating command, the number of digits of the S command exceeds the set value of parameter No.3031 or the commanded value is out of range.	Please modify the program or parameter No.3031.
066	The percentage in the spindle speed floating command is out of range.	In the spindle speed floating command, the percentage is not specified or the commanded percentage value is out of range (0.1~100).	The program needs to be modified.
067	The P value in the spindle speed floating command is incorrect.	In the spindle speed floating command, P is not specified or the commanded value is out of range.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
070	Commanded commands that cannot be run under DNC	There are commands that cannot be run under DNC, such as G36/G37, G70~G73	The program needs to be modified.
071	DNC timeout.	DNC transmission failed, please check.	
075	An axis with an invalid axis type setting is commanded.	The axis type setting of the commanded axis is invalid. Check the settings of parameters No.1006#0 and No.1006#1 corresponding to this axis.	The program or parameters need to be modified.
076	A rotary axis that cannot execute interpolation commands is commanded.	A rotary axis is commanded in non-polar coordinate and cylindrical interpolation states, and in non-G00/G01 modes.	Please modify the program or check the settings of parameter No.1006#0 and parameter No.1006#1.
77	A hidden axis that is not allowed to be programmed is commanded.	The program commands a hidden axis that is set as not allowed to be programmed by parameter #3408#0.	The program or parameter No.3115#0 or parameter No.3408#0 needs to be modified.
080	The basic axis attribute setting in the plane is incorrect.	The axis attribute setting of the basic axis corresponding to the interpolation plane is invalid.	Please modify parameter No.1022.
081	A rotary axis is commanded in circular interpolation.	The axis attribute setting of the basic axis corresponding to the circular interpolation plane is incorrect.	Modify the program or check the settings of parameter No.1006#0 and parameter No.1006#1.
082	The arc radius R and I/J/K are not specified.	In circular interpolation, neither R nor I, J, K is specified.	The program or parameter No.3403#5 CIR needs to be modified.
083	The radius command value in circular interpolation is incorrect.	In circular interpolation, the end point is not on the arc specified by R. Please check parameter No.3403#4 RER.	The program needs to be modified.
084	Exceeding the radius difference range.	In circular interpolation, the difference between the distance from the start point to the center and the distance from the end point to the center exceeds the value set in the parameters.	The program or parameters need to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
085	Too many helical axis commands in helical interpolation.	In helical interpolation mode, more than 3 axes are specified as helical axes.	The program needs to be modified.
086	Data error in three-point arc command.	In the three-point arc command, the midpoint is not specified or the specified midpoint cannot form an arc.	The program needs to be modified.
087	A full circle cannot be machined with three-point arc command.	A full circle cannot be machined with the three-point arc command, and the end point must be specified in the command.	The program needs to be modified.
088	Data error in three-point arc command.	1. In the three-point arc command, the specified midpoint and end point are collinear with the start point, or the start point is the same as the end point. 2. The radius of the arc specified by the end point and midpoint in the arc is 0.	The program needs to be modified.
096	The P or X/U of the dwell time in G04 command is out of range.	The value of P for the specified dwell time is out of the range of 0~9999999, or the value of X/U is out of the range of 0~9999.999.	The program needs to be modified.
100	In the thread cutting command, the J value of the retract length is incorrectly commanded.	In thread cutting, the specified J value exceeds the allowable range.	The program needs to be modified.
101	In the thread cutting command, the K value of the retract length is incorrectly commanded.	In thread cutting, the specified K value exceeds the allowable range.	The program needs to be modified.
102	The number of thread starts L is out of range.	The number of thread starts L specified in the thread command is out of the allowable range (1~99).	The program needs to be modified.
103	The thread lead command is out of range.	The lead F value is not within the required range.	The program needs to be modified.
104	The R command in the variable lead thread is out of range.	The lead F value is not within the required range, or in variable lead thread cutting, the lead variation specified by R is out of range.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
105	In the thread cutting command, the retract length of the major axis is too large.	The retract length of the major axis exceeds the thread cutting length.	The program needs to be modified.
106	In the thread cutting command, the retract length of the minor axis is too large.	The retract length of the minor axis exceeds the distance between the start point and the end point.	The program needs to be modified.
107	An axis outside the plane is commanded in the thread command.	In the thread command, an axis outside the selected plane is specified.	The program needs to be modified.
108	An axis that cannot be interpolated is commanded in the thread command.	In the thread command, the basic axis and parallel axis are commanded simultaneously, or more than two parallel axes are commanded, which cannot be interpolated.	The program needs to be modified.
109	The C-axis does not exist when the rigid thread command is issued.	When executing the rigid thread command, it is detected that there is no C address in parameter No.1020.	Parameter No.1020 is required.
110	The C-axis in the rigid thread command is not a rotary axis.	The C-axis in the rigid thread command is not a rotary axis. Please check the settings of parameter No.1006#0 and parameter No.1006#1.	The parameters need to be modified.
111	No spindle speed S is specified in the rigid thread command.	The spindle speed S value is not specified in the rigid thread command G32.1 line.	The program needs to be modified.
112	The minor axis feed distance (D) in the thread infeed/outfeed cutting cycle G92.2 command is incorrect.	The minor axis infeed distance value is less than or equal to 0.	The program or parameter No.5150 needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
113	The major axis feed distance (E) in the thread infeed/outfeed cutting cycle G92.2 command is incorrect.	The major axis infeed distance value is less than or equal to 0.	The program or parameter No.5151 needs to be modified.
114	The major axis feed distance (E) and outfeed distance (K) in the thread infeed/outfeed cutting cycle G92.2 command are incorrectly specified.	The sum of the major axis infeed distance and outfeed distance is greater than the thread machining length.	The program or parameters No.5151 and No.5153 need to be modified.
115	The minor axis outfeed distance (J) in the thread infeed/outfeed cutting cycle G92.2 command is incorrect.	The minor axis outfeed distance value is less than or equal to 0.	The program or parameter No.5152 needs to be modified.
116	The major axis outfeed distance (K) in the thread infeed/outfeed cutting cycle G92.2 command is incorrect.	The major axis outfeed distance value is less than or equal to 0.	The program or parameter No.5153 needs to be modified.
120	An axis outside the plane is specified in the circular thread command.	An axis outside the current plane is specified in the circular thread command.	The program needs to be modified.
121	No radius R and I/J/K are specified in the circular thread.	In circular thread interpolation, neither R nor I, J, K is specified.	The program or parameter No.3403#5 needs to be modified.
122	The radius command value in the circular thread is incorrect.	In circular thread interpolation, the end point is not on the arc specified by R. Please check parameter No.3403#4.	The program needs to be modified.
123	The radius difference between the start point and end point in the circular thread is out of range.	In circular thread interpolation, the difference between the distance from the start point to the center and the distance from the end point to the center exceeds the value set in parameter No.3410.	The program or parameters need to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
124	The R specified in the circular thread command is a negative value.	In the circular thread command, the value of R cannot be negative.	The program needs to be modified.
125	No arc end point is specified in the circular thread command.	In the circular thread command, no arc end point is specified, meaning the trajectory is a full circle which cannot be machined.	The program needs to be modified.
126	The start point and end point of the circular thread command are not in the same quadrant.	In the circular thread command, the specified start point and end point are not in the same quadrant, making machining impossible. The quadrants are defined as follows: first quadrant: 45°~135°; second quadrant: 135°~225°; third quadrant: 225°~315°; fourth quadrant: 315°~45°.	The program needs to be modified.
130	Illegal plane selection	In the plane selection command, two or more parallel axes are commanded in the same direction.	The program needs to be modified.
131	The basic axis of the current plane command is invalid		The program needs to be modified.
140	The G20/G21 command is not in the first line	The G20/G21 command must be in the first line of the program.	The program needs to be modified.
141	The G20/G21 commands for switching between inch and metric systems must be specified in separate lines.	The G20/G21 commands for switching between inch and metric systems cannot be in the same block as other G commands and must be specified in a separate line.	The program needs to be modified.
142	The G20/G21 commands for switching between inch and metric systems cannot be commanded in subprograms.	Inch/metric switching was executed when calling a subprogram.	The program needs to be modified.
150	The G22 command includes codes that cannot be in the same block.	The G22 command cannot be in the same block as other G commands and MSTF commands.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
151	The address data commanded by G22 exceeds the allowable value range.	The data input exceeds the allowable value range or the specified data exceeds 8 digits.	The data needs to be modified.
152	The stroke check range setting for G22 is incorrect.	In the stroke check range set by the G22 command, the specified positive coordinate value or parameter value is not greater than the negative coordinate value. Please check parameters No.1322 and No.1323.	The program needs to be modified.
153	The G22 command specifies an axis that cannot be commanded.	Axes other than X/Y/Z or U/V/W commands are specified in the G22 command.	The program needs to be modified.
154	The axis attribute setting for the G22 command is incorrect.	The basic axis attribute corresponding to the commanded X/Y/Z is set to 0 or is a parallel axis.	Please modify parameter 1022 ASA or modify the program.
160	G25/G26 cannot be in the same block as other G commands.	Other groups of G commands are simultaneously specified in the G25/G26 block.	The program needs to be modified.
165	The referenced point of the commanded axis has not been established.	Before cycle start, the reference point of the commanded axis has not been established.	Please establish the reference point or modify parameter No.1005#0 ZRNx.
166	G28 is commanded when the reference point has not been established.	The reference point has not been established before executing G28.	Please modify the program or parameter No.1002#3 AZR.
167	G30 or G53 is commanded when the reference point has not been established.	The reference point has not been established before executing G30 or G53.	Please establish the reference point.
168	An incorrect reference point P is commanded in G30.	In the block of G30 command, P specifies a value other than 2~4.	The program needs to be modified.
169	The axis of the intermediate point in G28/G30 command is out of range.	When returning to the reference point with G28/G30, the commanded intermediate point is out of the allowable range.	

Alarm No.	Significance	Possible Causes	Handling Method
180	An illegal L/P is commanded in G10.	1. In the program input offset (G10), neither the L value nor the P value is specified. 2. In the program input offset (G10), the specified L value or P value is not within the applicable range.	The program needs to be modified.
181	Programmable data input command does not match	The G10/G11 commanded in the program do not match, and the command G10 is repeated, or there is no command G10 before the command G11.	The program needs to be modified.
182	Programmable data entry is not canceled	Before the end of the program, G11 is not used to cancel the programmable data input mode.	The program needs to be modified.
183	Illegal command in programmable data input	NC commands such as axis address, G code or MSF are commanded in the programmable data input.	The program needs to be modified.
184	The parameter write command in programmable data input is illegal.	In programmable data input parameter writing mode, values other than P/R are commanded.	The program needs to be modified.
185	The parameter number P command for parameter writing in programmable data input is incorrect.	In programmable data input parameter writing mode, the parameter number P (0~9999) is not specified or the specified value is out of range.	The program needs to be modified.
186	The axis data Q for parameter writing in programmable data input is out of range.	In programmable data input parameter writing mode, the commanded axis number Q is out of range (1~maximum controlled axis number).	The program needs to be modified.
187	The bit data B command for parameter writing in programmable data input is incorrect.	In programmable data input parameter writing mode, when the parameter value R is not specified, the bit parameter number B is specified or the value of the specified bit number B is out of range (0~7).	The program needs to be modified.
188	The parameter value R for parameter writing in programmable data input is out of range.	In programmable data input parameter writing mode, the value of the commanded parameter B is out of range. When the bit parameter B is commanded, the value specified by parameter R can only be 0 or 1.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
196	The G10.9 command includes codes that cannot be in the same block.	The G10.9 command cannot be in the same block as other G commands and MSTF commands.	The program needs to be modified.
197	The G10.9 command is incorrect.	The command (G10.9) must specify at least one axis using absolute coordinates, and the commanded value can only be 0 or 1.	The program needs to be modified.
200	The commanded value of P in G31 is out of range.	The P value range in the G31 multi-step skip function is 1~12, 98 and 99.	The program needs to be modified.
201	Too many axes are commanded in the G31 skip command.	In the G31 skip command, only one of the basic axis and its corresponding parallel axis can be commanded.	The program needs to be modified.
202	G31 cannot be used in tool tip radius compensation mode.	A skip cutting command is specified in the tool tip radius compensation mode.	The program needs to be modified.
203	The Q value command in the G31 limit torque skip command is incorrect.	The multiplier value Q in G31 limit torque is out of range (1~300).	The program needs to be modified.
204	Too many axes are commanded in the G31 limit torque command.	Only one axis can be commanded in the G31 limit torque command.	The program needs to be modified.
205	The K value in the G31 command is out of range.	The K value commanded in the G31 skip function is out of range (allowable values are 0 and 1).	The program needs to be modified.
210	The automatic tool compensation G36/G37 command is invalid.	The automatic tool compensation function (G36, G37) is invalid.	The need to modify the program or modify the parameters No.6240#7 IGA
211	No offset number is found in automatic tool compensation G36/G37.	No tool offset number is specified before using the G36, G37 automatic tool compensation function.	The program needs to be modified.
212	T code is not allowed in automatic tool compensation G36/G37.	The automatic tool compensation G36, G37 commands cannot be in the same block as the T code.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
213	The axis commanded in automatic tool compensation G36/G37 is invalid.	The axis corresponding to the G36, G37 commands is an invalid axis.	Please modify parameter No.1022.
214	An illegal axis command is given in automatic tool compensation G36/G37.	In automatic tool compensation G36/G37, the axis of the movement command is not the axis corresponding to G36/G37 or the movement command is an incremental command.	The program needs to be modified.
215	The parameter setting for automatic tool compensation G36/G37 is incorrect.	The γ setting value is less than the ε setting value. Check the settings of parameter No.6251 ATOR1, parameter No.6254 ATOE1, parameter No.6252 ATOR2 and parameter No.6255 ATOE2.	The parameters need to be modified.
216	Automatic tool compensation G36/G37 cannot be used in tool tip radius compensation mode.	In the tool tip radius compensation mode, the automatic tool compensation function (G36, G37) command is specified.	The program needs to be modified.
230	The cylindrical interpolation G7.1 block contains commands that cannot be in the same block.	In the block commanding the cylindrical interpolation command G7.1, a G command from a different group or an MST command is specified.	The program needs to be modified.
231	The axis commanded when starting cylindrical interpolation is not a rotary axis.	The axis commanded in the block of G7.1 is not a rotary axis.	The program needs to be modified.
232	Too many rotary axes are commanded when starting cylindrical interpolation.	Too many rotary axes are commanded in the block of G7.1.	The program needs to be modified.
233	The radius commanded when starting cylindrical interpolation is incorrect.	The cylindrical radius commanded in the block of G7.1 is negative or not specified.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
234	G12.1 or G51.2 is commanded in cylindrical interpolation mode.	In cylindrical interpolation mode, the polar coordinate interpolation command G12.1 or polygon machining command G51.2 is commanded.	The program needs to be modified.
235	A plane switching command is commanded in cylindrical interpolation mode.	In cylindrical interpolation mode, G17~G19 cannot be commanded for plane switching.	The program needs to be modified.
236	A workpiece coordinate system selection command is commanded in cylindrical interpolation mode.	In cylindrical interpolation mode, G54~G59 cannot be commanded for workpiece coordinate system switching.	The program needs to be modified.
237	A multiple cycle command is commanded in cylindrical interpolation mode.	In cylindrical interpolation mode, multiple cycle commands such as G70~G78 cannot be commanded.	The program needs to be modified.
238	A tapping/drilling cutting command is commanded in cylindrical interpolation mode.	In cylindrical interpolation mode, tapping and drilling cycle cutting commands such as G84~G89 cannot be commanded.	The program needs to be modified.
239	A fixed cycle command is commanded in cylindrical interpolation mode.	In cylindrical interpolation mode, fixed cycle cutting commands such as G90~G94 cannot be commanded.	The program needs to be modified.
240	A thread command is commanded in cylindrical interpolation mode.	In cylindrical interpolation mode, thread cutting commands G32 and G34 cannot be commanded.	The program needs to be modified.
241	An axis outside the plane is commanded in cylindrical interpolation mode.	An axis outside the plane is commanded in cylindrical mode.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
242	A G code from group 00 other than G04 is commanded in cylindrical interpolation mode.	In cylindrical interpolation mode, G codes from group 00 other than G04 are commanded, including G27~G30, G31, G36/G37, G52 and G53.	The program needs to be modified.
243	A G00 command or T command is commanded in cylindrical interpolation mode.	In cylindrical interpolation mode, the rapid traverse command G00 or T command cannot be commanded.	The program needs to be modified.
245	The cylindrical interpolation command G7.1 cannot be commanded in C tool compensation mode.	In C tool compensation mode (non-G40 modal), the cylindrical interpolation command G7.1 is commanded.	The program needs to be modified.
246	The K value in cylindrical interpolation mode is incorrect.	The K value commanded in the block of G7.1 is out of the allowable range (0-1).	The program needs to be modified.
250	The axis type corresponding to the polar coordinate interpolation axis parameter setting is incorrect.	The axis type (parameters No.1006#0 and No.1006#1) corresponding to the axis set in polar coordinate interpolation axis specification parameter No.5460 (linear axis) or parameter No.5461 (rotary axis) is set incorrectly. The axis type corresponding to the axis set in parameter No.5460 should be a linear axis. The axis type corresponding to the axis set in parameter No.5461 should be a rotary axis.	Please modify parameters No.5460 and No.5461, or parameters No.1006#0 and #1.
251	The polar coordinate interpolation command is commanded in C tool compensation mode.	In C tool compensation mode (non-G40 modal), polar coordinate interpolation commands G12.1/G13.1 are specified.	The program needs to be modified.
252	An axis outside the plane is commanded in the arc command under polar coordinate interpolation mode.	In polar coordinate interpolation mode, the axis of the arc command is not in the selected plane.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
253	The polar coordinate interpolation commands G12.1/G13.1 cannot be in the same block as G commands from other groups.	G commands from other groups are commanded in the block of polar coordinate interpolation commands G12.1/G13.1.	The program needs to be modified.
254	The polar coordinate interpolation command G12.1 is repeated.	In polar coordinate interpolation mode, the G12.1 command is specified again.	The program needs to be modified.
255	A T code is commanded in polar coordinate interpolation mode.	In polar coordinate interpolation mode, T commands cannot be executed.	The program needs to be modified.
256	An unavailable command is specified under polar coordinate interpolation.	In polar coordinate interpolation mode, G codes that cannot be used are specified. Except for the following G codes, other G codes are unavailable: 1. G04 and G65 in group 00; 2. G01, G02 and G03 in group 01; 3. G98/G99 for A set of G codes and G94/G95 for B set of G codes in group 03; 4. G40~G43 in group 05 and G66/G67 in group 09.	The program needs to be modified.
270	The G50 command cannot be used.	Parameter 1202#2 is set to 1, which means an alarm is generated when specifying the coordinate system setting command (G50 for A set of G codes, G92 for B set of G codes).	The program or parameters need to be modified.
271	The coordinate system setting value is out of range.	The coordinate system values set by G50 (G92 for B set of G codes)/G52/G53/G54~G59 are out of range.	
272	The P value of the additional workpiece coordinate system is out of range.	When commanding the additional workpiece coordinate system G54.1, the coordinate system selection command P is out of range.	Please modify the program.
275	A non-zero value is commanded for the axis specified in the workpiece coordinate system preset command.	In the workpiece coordinate system preset commands G50.3/G92.3, the specified axis commands values other than 0.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
280	The subprogram call address P is undefined.	In the program with M98, M198, G65 or G66 commands, the address P (program number) is not specified.	The program needs to be corrected.
281	The number of subprogram call layers is excessive.	The subprogram call exceeds 12 levels.	The program needs to be modified.
282	The N number commanded by address P is not found.	The program number or sequence number specified by address P is not found in the block containing M98, M198, M99, G65 or G66.	The program needs to be modified or the background editing needs to be interrupted.
283	Failed to open the program when calling the subprogram.	When calling a subprogram and prereading the subprogram internally, the subprogram fails to open.	Reset or restart the program after power-on.
284	Subprogram buffer initialization failed.	When calling a subprogram, the initialization of the subprogram buffer fails.	It is necessary to reset or restart after power-on.
285	Subprogram call error occurred.	The M98, M198, G65 or G66 command calls the superior program or itself.	The program needs to be modified.
286	The called subprogram is in use.	The called subprogram is being edited or has been edited but not saved.	Please save the program and reset before running again.
287	Program call statements cannot run in MDI and DNC modes.	The system does not support calling itself in MDI mode and calling macro programs and subprograms in DNC mode.	The program needs to be modified.
288	Only one layer of subprogram can be called in MDI mode.	Only one level of subprogram can be called in MDI mode.	The program needs to be modified.
289	The file name specified by P in the subprogram call is incorrect.	When calling with P[], only macro variables or character strings can be placed in "[]".	The program needs to be modified.
290	The data of L (subprogram call times) is incorrect.	The commanded value of L (subprogram call times) or the data corresponding to the macro variable of the L command is incorrect (it cannot be less than 1 or greater than 9999).	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
291	The value of the macro variable in the P command for subprogram call is incorrect.	The value corresponding to the macro variable using the P command is empty or out of range.	The program needs to be modified.
292	The commanded value of the Q command in subprogram call is incorrect.	In M98 and M198 subprogram calls, the line number N of the Q command is out of range.	The program needs to be modified.
293	The subprogram call times are commanded repeatedly.	When calling M98 and M198 subprograms, the number of repeated calls is commanded via P, and L is also commanded at the same time.	The program needs to be modified.
294	Q is not commanded when a subprogram calls itself.	When M98 and M198 subprograms call the current program, the call line number Q is not commanded.	The program needs to be modified.
300	When selecting multiple spindles, G commands that cannot be in the same block are commanded.	In the spindle selection function based on address P in multi-spindle control, G codes that cannot be commanded simultaneously with S_P_ commands are specified. S_ cannot be in the same block as G25/G26, and S_P_ cannot be in the same block as G96/G97.	The program needs to be modified.
301	The spindle selection P command is out of range.	In the spindle selection function based on address P in multi-spindle control, the commanded P value is out of range (1~maximum number of spindles).	The program or parameters need to be modified.
302	The multiple spindle selection function is invalid.	When parameter 3710 is set to 1 or parameter 3703#3=0, the spindle selection P_S_ command is commanded.	The program or parameters need to be modified.
303	When using the S command for multiple spindle selection, the axis number command is incorrect.	When using the Sn=_ format command for multi-spindle selection, the axis number n of the S command exceeds the maximum number of spindles.	The program or parameters need to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
304	The multiple spindle selection P command is not specified.	In the spindle selection function based on address P in multi-spindle control, when parameter 3706#2 is set to 0, there is no P command following the spindle speed S.	The program or parameters need to be modified.
305	The parameter setting is incorrect when the multiple spindle selection function is based on P selection.	When the multi-spindle function is valid and selection is based on address P, the spindle number set in parameter 3775 or parameter 3776 is greater than the maximum number of spindles.	The parameters need to be modified.
310	The polygon machining command is commanded in C tool compensation mode.	In C tool compensation mode (non-G40 modal), the polygon machining command G51.2 is specified.	The program needs to be modified.
311	The P command in polygon machining G51.2 is incorrect.	In the block of G51.2, P is not specified or the specified P value is out of range.	Please modify the program
312	The Q command in polygon machining G51.2 is incorrect.	In the block of G51.2, Q is not specified or the specified Q value is out of range.	Please modify the program
313	Commands that cannot be in the same block are specified in the polygon machining G51.2 or G50.2 block.	Polygon machining G51.2/G50.2 must be commanded independently and cannot be in the same block as other commands.	Please modify the program
314	The commanded axis is illegal in polygon machining.	During polygon machining, a movement command for the tool rotation axis is commanded.	Please modify the program
315	The S command is not commanded when the workpiece axis is in position mode during polygon machining.	In polygon machining, when the workpiece axis is in position mode (parameter No.7603#4=1), the S command is not commanded.	Please modify the program or parameter No.7603#4.
316	An illegal command is given in polygon machining.	During polygon machining, commands such as G7.1, G12.1, G51.3 are commanded.	Please modify the program

Alarm No.	Significance	Possible Causes	Handling Method
317	The polygon machining G51.2 or helical tooth interpolation G51.3 is commanded in fixed cycle mode.	When the fixed cycle command is valid, polygon machining G51.2 or helical tooth interpolation G51.3 cannot be commanded.	Please modify the program
318	The workpiece spindle axis number D command in the polygon machining G51.2 block is incorrect.	The workpiece spindle axis number D commanded in the polygon machining command block is out of range.	Please modify the program
319	The tool rotation axis number E command in the polygon machining G51.2 block is incorrect.	The tool rotation axis number E commanded in the polygon machining command block is out of range.	Please modify the program
320	An axis number command is commanded in polygon machining G51.2 mode.	In polygon machining mode, the workpiece axis number D or tool rotation axis number E is commanded.	Please modify the program
321	The starting angle R command in the polygon machining G51.2 program is out of range.	In polygon machining mode, the commanded value of the starting angle R is out of range.	Please modify the program
330	The G35 command contains codes that cannot be in the same block.	The G35 command cannot be in the same block as other G commands or M/T commands.	The program needs to be modified.
332	The torque command in the G35 command is illegal.	No torque is commanded in the G35 command or the commanded torque is less than 0.	The program or parameters need to be modified.
333	The axis command in the G35 command is incorrect.	The torque command cannot be an incremental command or more than two torque commands are commanded.	The program needs to be modified.
334	The F value commanded in the G35 command is out of range.	The F value commanded in the G35 command is out of range (0.001~99999).	The program needs to be modified.
335	The S and F commands are not specified in the G35 command.	Neither the S command nor the F command is commanded in the G35 command.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
336	Both J and K are commanded in the G35 command.	The alarm margin J and alarm flag K cannot be commanded simultaneously in the G35 command.	The program needs to be modified.
337	The K value in the G35 command is incorrect.	The K value in the G35 command can only be 0 or 1.	The program needs to be modified.
338	Both I and P are commanded in the G35 command.	The torque incremental command I and absolute command P cannot be commanded simultaneously in the G35 command.	The program needs to be modified.
340	The P value of the parabola G2.2/G3.2 command is incorrect.	In the parabola G2.2/G3.2 command, P is not commanded or the commanded P value is zero, which cannot form a parabola.	The program needs to be modified.
341	The end point of the parabola G2.2/G3.2 command is incorrect.	In the parabola G2.2/G3.2 command, the straight line where the start point and end point lie is parallel to the axis of symmetry, which cannot form a parabola, or the calculated vertex is out of range.	The program needs to be modified.
342	The data of the parabola G2.2/G3.2 command cannot form a parabola.	The data given by the parabola G2.2/G3.2 cannot form a correct curve segment or lacks necessary command words.	The program needs to be modified.
343	The Q value of the parabola G2.2/G3.2 command is incorrect.	The Q (angle value) of the parabola G2.2/G3.2 command is out of the allowable range.	The program needs to be modified.
351	The radius/major axis I value commanded in eccentric circle G6.4 or elliptical cylinder G6.5 interpolation is incorrect.	In the G6.4/G6.5 command, the radius/major axis I has no commanded value or the commanded value is out of range.	The program needs to be modified.
352	The minor axis J value commanded in elliptical G6.5 interpolation is incorrect.	In the G6.5 command, the minor axis J has no commanded value or the commanded value is out of range ($J > 0$).	The program needs to be modified.
353	The rotary axis setting in eccentric circle G6.4, ellipse G6.5, and polygon G6.6 interpolation is incorrect.	The axis set in parameter No.5360 is not a rotary axis. Modify parameter No.5360 or parameters No.1006#0 and No.1006#1.	The program or parameters need to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
354	In eccentric circle G6.4, ellipse G6.5, and polygon G6.6 interpolation, no linear axis is commanded.	In the G6.4/G6.5/G6.6 commands, the value of the linear axis set by parameter 5361 is not commanded.	The program or parameters need to be modified.
355	In ellipse G6.5 interpolation, the values of major and minor axes I and J are incorrectly commanded.	In the G6.5 command, the value of the major axis I is smaller than the value of the minor axis J.	The program needs to be modified.
356	In eccentric circle G6.4, ellipse G6.5, and polygon G6.6 interpolation, the rotational speed S of the rotary axis is not commanded.	In the G6.4/G6.5/G6.6 commands, the rotational speed S of the rotary axis is not commanded.	The program needs to be modified.
357	In eccentric circle G6.4, ellipse G6.5, and polygon G6.6 interpolation, the rotation command is incorrect.	In the G6.4/G6.5/G6.6 block, the rotary axis set by parameter No.5360 cannot be commanded.	The program needs to be modified.
358	In eccentric circle G6.4 or elliptical cylinder G6.5 interpolation, the value of eccentricity K is incorrectly commanded.	In the G6.4/G6.5 commands, the value of eccentricity K is out of range or larger than the value of radius (or major axis) I.	The program needs to be modified.
359	In eccentric circle G6.4, elliptical cylinder G6.5, and polygon G6.6 interpolation, too many linear axes are commanded.	In the G6.4/G6.5/G6.6 commands, two or more linear axes other than the axis set by parameter No.5361 are commanded.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
360	In eccentric circle G6.4 or elliptical cylinder G6.5 interpolation, the positioning point is incorrect.	When commanding G6.4 or G6.5, for internal hole machining, the absolute coordinate value of the positioning point must be less than the value of (I+K); for external hole machining, the absolute coordinate value of the positioning point must be greater than the value of (I+K).	The program needs to be modified.
361	In polygon cutting, the number of sides L is incorrectly commanded.	In the polygon cutting G6.6 command, the number of sides L is not commanded, or the commanded L is out of range (only 4 or 6 are allowed).	The program needs to be modified.
370	In the coordinate rotation G68.1 program, commands that cannot be in the same block are commanded.	In the G68.1 block, G commands or MST commands that cannot be in the same block are commanded.	The program needs to be modified.
371	The axis commanded in the coordinate rotation G68.1 program is incorrect.	In the G68.1 block of the program, the axis commanding the rotation center is not the axis of the current interpolation plane.	The program needs to be modified.
372	An unauthorized G code is commanded in coordinate rotation G68.1 mode.	In coordinate rotation mode, only G00, G01, G83~G89 commands can be commanded.	The program needs to be modified.
373	The K value commanded in the coordinate rotation G68.1 block is incorrect.	In the G68.1 block of the program, the commanded value of K can only be 0 or 1.	The program needs to be modified.
374	G68.1 cannot be commanded again in coordinate rotation G68.1 mode.	In the G68.1 modal state of the program, G68.1 is commanded again.	The program needs to be modified.
380	A rotary axis is commanded in ellipse G2.1 or ellipse G3.1 interpolation.	In ellipse interpolation, a rotary axis is commanded. Modify the program or check the settings of parameter No.1006#0 and parameter No.1006#1.	The program or parameters need to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
381	The value of the major axis I specified for ellipse G2.1 or ellipse G3.1 is incorrect.	The value of the major axis I specified for ellipse G2.1 or ellipse G3.1 is out of range.	The program needs to be modified.
382	The value of the minor axis J specified for ellipse G2.1 or ellipse G3.1 is incorrect.	The value of the minor axis J specified for ellipse G2.1 or ellipse G3.1 is out of range.	The program needs to be modified.
383	The data specified for ellipse G2.1 or ellipse G3.1 cannot form an ellipse.		The program needs to be modified.
384	A full circle cannot be commanded with ellipse G2.1 or ellipse G3.1.	The starting coordinates and ending coordinates of ellipse G2.1 or ellipse G3.1 are the same.	The program needs to be modified.
385	The end point commanded for ellipse G2.1 or ellipse G3.1 is incorrect.	The distance between the end point and the start point commanded by ellipse G2.1 or ellipse G3.1 is greater than the value of the major axis I.	The program needs to be modified.
386	The angle commanded for ellipse G2.1 or ellipse G3.1 exceeds the allowable range.		The program needs to be modified.
400	The parameter switch is turned on.		Press the [Reset] key to clear the alarm.
401	Failure to open the part program.		The current program has been removed or does not exist; please select the program again.
402	Parameter backup failed.		Please check the memory or restart the power and try again.
403	Parameter restoration failed.		Please check if parameters are being written, or restart the power and try again.

Alarm No.	Significance	Possible Causes	Handling Method
405	System parameters one-click recovery succeeded.		Please restart the power.
406	PLC parameters one-click recovery succeeded.		Press the Reset key to clear the alarm.
407	Servo parameters one-click recovery succeeded.		Please restart the power.
408	I/O unit parameters one-click recovery succeeded.		Please restart the power.
409	The input parameter file is invalid.	The parameter file input externally is invalid; the original data has been restored.	Please re-enter the parameter file.
410	Operation error in program scheduling mode.	Running programs in program scheduling mode is prohibited except in automatic mode. Please reset the operation first to clear the program scheduling mode.	
411	Custom configuration (machine tool diagnosis/GSKUI/parameters/macro variables/startup picture) one-click recovery succeeded.		Press the Reset key to clear the alarm.
412	CNC program one-click recovery succeeded.		Press the Reset key to clear the alarm.
413	Part program loading failed.		Press the Reset key to clear the alarm, then reselect and load the program.
414	MDI subprogram loading failed.	MDI cannot call encrypted subprograms.	Press the Reset key to clear the alarm, then reselect and load the program.

Alarm No.	Significance	Possible Causes	Handling Method
415	Multiple PLC data exceed the set range.	More than 3 PLC data items exceed the set range. This is because the PLC data are not within the set range due to system upgrade or PLC switch without overwriting with default data.	Please check on the PLC data page.
416	Additional program opening failed.		The current additional program has been removed or does not exist; please reselect the additional program.
430	Multiple parameter data exceed the set range.	More than 3 parameter data items exceed the set range; default values have been used.	Please check the parameters on the parameter page.
450	Parameters have been modified; please restart the power.	.	After the parameter is input, it will take effect after power-on again.
451	Servo parameters have been modified; please restart the power for both the system and the servo.		After some servo parameters are modified, both the system and the servo must be completely powered off and restarted.
452	The bus communication station address has been modified; this device must be restarted.		Please power off and restart this slave station; otherwise, the modified station address will not take effect.
453	I/O unit parameters have been modified; please restart the power.		After inputting I/O unit parameters, the power must be restarted for them to take effect.
460	The number of CNC control axes is greater than the total controlled axes; the system has stopped.		Please check parameters No.1010 and No.8130.
461	Identical axis attributes have been set; the system has stopped.		Please modify parameter No.1022.
462	The axis name setting is incorrect.	Possible causes include: 1. Duplicate axis names have been set; 2. Axis names prohibited under the current G-code system have been specified.	Please modify parameter No.1020, 8412, or parameter No.3401#6.

Alarm No.	Significance	Possible Causes	Handling Method
463	The rotary axis setting is invalid; the system has stopped.	The rotary axis setting in parameter No.1006 is invalid.	Please modify parameter No.1006.
465	The control axes have been set with the same communication station address.		Please modify parameter No.1023.
466	The master axis setting in the electronic gearbox function is incorrect.	1. The set axis number exceeds the total number of controlled axes; 2. The set axis is the axis itself; 3. The set axis is both a master axis and a slave axis; 4. Multiple groups of master/slave axes have been set.	Please modify parameter No.7710.
467	The axial feed axis number in the helical interpolation function is not specified.	1. The set axis number exceeds the total number of controlled axes; 2. Neither the axial feed axis number is directly specified nor the default Z-axis is enabled.	Please modify parameter No.7711 to a non-zero value or enable the Z-axis (axis attribute 3) in parameter No.1022.
468	The axial feed axis number setting in the helical interpolation function is incorrect.	The set axis number cannot be the master axis or slave axis in the electronic gearbox function.	Please modify parameter No.7711 or parameter No.7710.
469	The master axis number setting in the feed axis synchronization control is incorrect.	1. The set axis number exceeds the total number of controlled axes, or the set axis is both a master axis and a slave axis; 2. There is a difference between the master axis and the slave axis in terms of linear/rotary axes.	Please modify parameter No.1006#0, No.1006#3, or No.8130.
470	The spindles have been set with the same communication station address or analog output address.		Please modify parameter No.3717.
471	The I/O unit corresponding to the analog output address mapped by the spindle cannot be found.		Please modify parameter No.3717.

Alarm No.	Significance	Possible Causes	Handling Method
472	The spindle encoder channel numbers are the same.		Please modify parameter No.3723.
473	The axis with Cs Cs contour control function enabled is not within the range of NC axes.		Please check parameters No.1010 and No.8130.
474	No valid spindle station address is set for Cs contour control.		Please modify parameters No.3717 and No.3701#7.
475	For Cs contour control, the station addresses of the spindle and feed axis or auxiliary axis are not set to be consistent.		Please modify parameters No.1023, No.8414, No.3717, and No.3701#7.
476	The station addresses of the spindle and feed axis or auxiliary axis are set to be consistent, but the Cs contour control function is not enabled.		Please modify parameters No.1023, No.8414, No.3717, and No.3701#7.
477	The analog output address mapped by the spindle exceeds the maximum number of AO terminals of the corresponding I/O unit.		Please modify parameter No.3717.
478	For Cs contour control, the spindle and feed axis do not point to the same pulse I/O unit.		Please modify parameters No.1023, 3717, or 3701#7.
479	The master axis number setting in spindle synchronization control is incorrect.		Please modify parameter No.4831.

Alarm No.	Significance	Possible Causes	Handling Method
480	The I/O units are set with the same communication station address.		Please modify parameters No.3051, No.3052, No.3053, and No.3054.
485	The gateway is set with the same communication station address.		Please modify parameters No.3061 and No.3062.
490	No valid communication station address is set.		Please modify parameter No.9000#0 or reset parameters No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, No.3060, No.3061 or No.3062.
491	The macro variable configuration has been changed; please restart the power.		The macro variable configuration file of the current system has been changed and will take effect only after restarting the power.
492	There is an error in the user-customized macro variable interface configuration.	The system has enabled the user-customized interface function, but no valid configuration file was found.	Please modify parameter 8132#6 or execute the customized macro variable configuration file on the macro variable list page.
493	The I/O unit corresponding to the pulse axis terminal mapped by the control axis cannot be found.		Please modify parameter No.1023.
494	The pulse axis terminal mapped by the control axis exceeds the maximum number of pulse axis terminals of the corresponding I/O unit.		Please modify parameter No.1023.
495	The auxiliary axes are set with the same communication station address.		Please modify parameter No.8414.

Alarm No.	Significance	Possible Causes	Handling Method
496	The auxiliary axis and feed axis are set with the same communication station address.		Please modify parameters No.1023 and No.8414.
500	The reference point has not been established.	Please manually move to the reference point, then press the axis movement key in the reference point return mode to establish the reference point.	
501	Encoder data error.	Encoder data reading error.	The machine reference point needs to be re-established.
504	Encoder battery undervoltage alarm.	This alarm occurs for the following two reasons: 1. The encoder battery voltage is lower than the minimum normal operating voltage; 2. Poor contact of the encoder cable or the encoder cable has been plugged/unplugged.	The battery needs to be replaced or the encoder cable connection needs to be checked for good contact. Note: After the alarm is cleared, the system needs to re-establish the zero coordinate.
510	Alarm for large deviation of machine coordinate system upon power-on.	The error value between the machine coordinate established upon power-on and the memorized machine coordinate exceeds the allowable tolerance. Reasons: 1. The carriage position moved during machine power-off; 2. The set value of parameter No.1206 is too small.	The machine reference point needs to be reset or the power needs to be restarted.
511	Alarm for excessive deviation of machine coordinate system upon power-on.	The error value between the machine coordinate established upon power-on and the memorized machine coordinate is too large. Reasons: 1. The carriage position moved during machine power-off; 2. The motor encoder has been replaced.	Please restart the power or reset the machine reference point.
512	System parameters related to reference point have been changed.	Power-on detection found that the system parameters are different from those when the machine reference point was established, including parameters No.1811#2, No.1816, or No.1820.	The machine reference point needs to be re-established.

Alarm No.	Significance	Possible Causes	Handling Method
513	Servo parameters related to reference point have been changed.	Power-on detection found that the servo parameters are different from those when the machine reference point was established, including command inversion parameters or gear ratio parameters.	The machine reference point needs to be re-established.
514	Failed to read parameter record file related to reference point.	Power-on detection failed to find the parameter values from the last successful establishment of the machine zero point.	The machine reference point needs to be reset.
520	PC signal detection unsuccessful due to unconnected drive unit or bus disconnection.		Please check the bus connection or restart the power and try again.
521	Torque control switching error.	The reference point of the absolute encoder for this axis is not established during torque switching.	Please establish the reference point first.
522	Excessive mechanical coordinate synchronization error.	During the synchronous operation of feed axis synchronization control, the mechanical coordinate difference between the master axis and the slave axis exceeds the set value of parameter No.8314.	Please modify the parameters or perform synchronization adjustment.
530	Absolute coordinates inconsistent upon power-on.	The absolute coordinates are inconsistent with those at power-off, caused by the system canceling the G50 setting value.	Please check the coordinates or modify parameter 1201#4.
604	Servo alarm.	Digital servo system failure.	Please check the servo.
650	Servo operation power-off alarm.	The servo was powered off while executing a movement command, which may result in incorrect coordinate positions.	Please return to the reference point again.
700	Positive travel limit 1	Moving in the positive direction exceeded stored stroke detection 1.	Please modify parameter No.1320.
701	Negative travel limit 1	Moving in the negative direction exceeded stored stroke detection 1.	Please modify parameter No.1321.
702	Positive travel limit 2	Moving in the positive direction exceeded stored stroke detection 2. Inside/outside detection is determined by parameter No.1300#0.	Please modify parameters No.1300, No.1322, or No.1323.

Alarm No.	Significance	Possible Causes	Handling Method
703	Negative travel limit 2	Moving in the negative direction exceeded stored stroke detection 2. Inside/outside detection is determined by parameter No.1300#0.	Please modify parameters No.1300, No.1322, or No.1323.
704	Positive travel limit 3	Moving in the positive direction exceeded stored stroke detection 3.	Please modify parameters No.1324 or No.1325.
705	Negative travel limit 3	Moving in the negative direction exceeded stored stroke detection 3.	Please modify parameters No.1324 or No.1325.
706	Positive overtravel	Moving in the positive direction exceeded the hard limit.	Please cancel the overtravel by canceling the limit or modify parameter No.3004.
707	Negative overtravel	Moving in the negative direction exceeded the hard limit.	Please cancel the overtravel by canceling the limit or modify parameter No.3004.
710	Spindle speed fluctuation detection alarm	In the spindle speed fluctuation detection function, the actual speed exceeds the commanded speed range.	Please check the machine connection and cutting conditions or modify parameters such as 4912 and 4913.
720	Manual tool measurement error	The measurement operation is illegal. Possible causes are: 1. The detection signal is input when two axes are moving simultaneously or when the axis is not moving; 2. A detection signal inconsistent with the axis movement direction is input during measurement; 3. The movement direction of the axis is not constant, making it impossible to determine the direction; 4. Other detection signals are input before exiting the measurement in a certain direction; 5. Manual tool setting is currently in progress.	Please re-measure.
721	The reference point is not established during manual tool measurement.	The reference point of the axis is not established.	Please establish the reference point first and re-measure.

Alarm No.	Significance	Possible Causes	Handling Method
722	Data writing error occurs during manual tool measurement.	Data writing error occurs during manual tool measurement.	Please check if the memory is functioning properly.
723	Error in selecting the tool offset number via PLC during manual tool measurement.	Error in selecting the tool offset number via PLC during manual tool measurement.	Please check if the ladder diagram is correct.
730	Alarm for exceeding the effective distance during manual tool measurement.	During manual tool measurement, the measurement input signal is not detected within the range set by parameter 5023.	Please modify the value of parameter 5023 or re-input the measurement after moving the tool.
736	X-axis positive direction is disabled.		
737	X-axis negative direction is disabled.		
738	Z-axis positive direction is disabled.		
739	Z-axis negative direction is disabled.		
740	Axis interlock signal is active.		
741	All axes interlock signal is active.		
742	Cutting block interlock signal is active.		
743	Block interlock signal is active.		
750	In-position width detection is in progress.		
751	Axis servo enable is disconnected.		
752	Axis operation mode error.		
753	PLC signal G261 is disabled.		
905	Auxiliary coordinate data creation failed.	G54.1 auxiliary coordinate data creation failed.	Press the reset button to clear the alarm, then restart the power or send it to the manufacturer for repair.

Alarm No.	Significance	Possible Causes	Handling Method
910	Error in initializing parameters.	The user parameter file does not exist or the data has been corrupted, and cannot be restored with default parameters. Factory parameters have been used.	
911	Error in initializing CNC configuration.	The CNC configuration file does not exist or the data has been corrupted, and default configuration has been used.	
912	Error in initializing tool offset data.	The tool offset file does not exist or the data has been corrupted, and cannot be restored with the default tool offset file. Initial data has been used.	
913	Error in initializing tool life data.	The tool life file does not exist or the data has been corrupted, and initial data has been used.	
914	Error in initializing lead screw compensation data.	The lead screw compensation file does not exist or the data has been corrupted, and initial data has been used.	
915	Error in initializing PLC program.	File reading failed or compilation error occurred during loading.	
919	System voltage is too low during power-on.	Please power off and then restart.	
920	System voltage is too low during operation.	The system voltage was too low during operation.	Please power off the device and then restart it.
930	Error in initializing tool offset data.	The tool offset file was corrupted or the verification failed, and has been restored using the backup file.	
931	Error in initializing parameter file.	The parameter file was corrupted or the verification failed, and has been restored using the backup file.	
932	Chinese input method creation failed.	The Chinese input method file is corrupted and cannot be used.	
940	NVRAM data changed.	The current NVRAM data version is inconsistent with the data version before the last power-off.	If the system uses an absolute encoder, please re-establish the reference point. For incremental encoders, please re-perform the zero return operation, re-execute the PLC, and overwrite the registers with the default data of the ladder diagram.

Alarm No.	Significance	Possible Causes	Handling Method
941	Power-off occurred while NVRAM was in operation.	The system lost power during operation.	Please re-perform the zero return operation and confirm whether the coordinates, tool offset values and status are correct.
942	NVRAM data area 1 is abnormal.	The checksum of NVRAM data area 1 does not match or it has been corrupted.	Please re-perform the zero return operation and confirm whether the coordinates, tool offset values and status are correct. If this alarm occurs frequently, please send the equipment to the manufacturer for maintenance.
943	NVRAM data area 2 is abnormal.	The checksum of NVRAM data area 2 does not match or it has been corrupted.	If the system uses an absolute encoder, please re-establish the reference point. For incremental encoders, please re-perform the zero return operation. If this alarm occurs frequently, please send the equipment to the manufacturer for maintenance.
950	The time-limited shutdown period has expired, and the system cannot work normally.		Please contact the sales staff.
951	Memory failure, important files lost, and the system cannot operate normally.		Please contact the system maintenance personnel.
952	The system memory resources are insufficient, and the system may not work normally.		Please try to reduce the number of axes or contact the system maintenance personnel.
990	Too many alarms or prompts.	The total number of alarms exceeds 20 or the total number of prompts exceeds 30.	

Alarm No.	Significance	Possible Causes	Handling Method
991	Unrecognizable alarm number.	The alarm content cannot be found by the alarm number.	
992	Data error in the alarm information.	In the alarm information or operation information, some data is incorrect.	
993	Data error in the PLC alarm information table.	In the PLC alarm information, there is no commanded PLC alarm number or it is outside the specified range of 1000~2999.	Please modify the ladder diagram display information table.
4000	Syntax check not completed due to reset.	During the syntax check, the reset button was pressed, causing the syntax check to be incomplete. Please check parameter 3401#2 NCK.	Please recheck the program.
4001	Duplicate N numbers found during syntax check.	During the syntax check, duplicate program block numbers appear, which may cause errors.	The program needs to be modified.
4009	System parameter No.1010 is set to 0, and no CNC control axes are enabled.	Press reset to clear this alarm.	
4010	The parameter value is not within the set range.	Due to system upgrade, ISB/ISC precision switching, or linear axis/rotary axis switching, the parameter is outside the set range. The parameter has been set to the default value.	
4011	GSK-Link is not enabled, but a valid communication station address has been set.		Please modify parameter No.9000#0, or reset parameters No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, No.3060.
4012	The user-customized macro variable interface configuration does not exist.	The system has enabled the user-customized interface function, but the configuration file was not found or the current configuration file does not contain configuration information.	Please modify parameter 8132#6 or execute the customized macro variable configuration file on the macro variable list page.

Alarm No.	Significance	Possible Causes	Handling Method
4013	The user-customized macro variable interface configuration is incorrect.	The system has enabled the user-customized interface function, and the configuration information in the configuration file is incorrect.	Please execute the correct macro variable configuration file.
4014	During temperature compensation, the position of the heating end point and the cooling start point in the thermal balance state are inconsistent.	In the temperature compensation function, the cumulative compensation value in the X-axis heating area is inconsistent with that in the cooling area.	Please modify the temperature compensation data.
4015	The user-customized interface has been updated.	The configuration file of the user-customized interface has changed.	Please power on again to check.
4016	The value of the register is not within the set range.	Due to system upgrade or switching PLC without overwriting with default data, the PLC data is outside the set range.	
4017	The machine tool diagnostic configuration file has been modified.	The machine tool diagnostic configuration file has changed.	Please power on again.
4018	The imported lead screw compensation data is out of range and has been clamped to the maximum value.	Some of the imported lead screw compensation data exceeds the system maximum value.	It is recommended to adjust the precision of the lead screw compensation import file and re-import it.
4020	The spindle encoder uses the GSK-Link channel by default, but no valid station address has been set for the spindle.		Please modify parameters No.3717 and No.3723 to non-zero values.
4021	User-defined parameters have been updated.	The configuration file of user-defined parameters has changed.	Please power on again to check.

Alarm No.	Significance	Possible Causes	Handling Method
4022	The startup picture has been updated.	The startup picture has changed.	Please power on again to check.
4023	The custom macro variable file has been updated.	The custom macro variable file has changed.	
4030		The shutdown operation password is the default password.	For safety reasons, please change the password.
4040	Service life has been exhausted.	Please replace the tool whose service life has been exhausted and reset the used life of the exhausted tool to zero.	Press Cycle Start to continue execution.
4041	Service life is about to be exhausted.	Please replace the tool whose service life is about to be exhausted as soon as possible.	
4100	FPGA parameter setting failed.	FPGA parameters have not been set successfully, which may affect thread processing or spindle control.	Please power on again or send the equipment to the manufacturer for maintenance.
4110	Encoder battery undervoltage warning.	The encoder battery voltage is lower than the warning voltage. Note: After this warning appears, the user can continue processing, but the battery needs to be replaced as soon as possible.	Please replace the battery. Note: Replace the battery when the drive unit displays the warning (i.e., when the servo drive unit is powered on), then power off. The warning will be cleared after powering on again.
4120	The slave station number of the gateway's GSK-Link-PA port has been modified.	The communication loop of the gateway's GSKLink-PA port must be powered on again.	The communication loop on the GSK-Link-PA side of the gateway must be powered on again.
4200	Error in obtaining the device number by the operation panel.		Please check the connection between the machine tool panel and the system.
4201	Error in obtaining device information by the operation panel.		Please check the connection between the machine tool panel and the system.

Alarm No.	Significance	Possible Causes	Handling Method
4202	Continuous errors occur during normal communication of the operation panel.		Please check the connection between the machine tool panel and the system.
4205	The soft panel function switch has been turned on.		Please modify system parameter No.8136#3 (NOP) to set whether the machine tool panel is used.
4210	Editing keyboard communication error.	Communication error between the editing keyboard and the system.	
4211	Keyboard code table communication error.	Communication error between the keyboard code table and the system.	
4300	The system is currently at level 1 permission.	Under system level 1 permission, please do not modify parameters arbitrarily.	
4305	The system is about to reach the shutdown time.		To avoid losses caused by shutdown, please contact the sales staff to lift the restriction in advance.
4306	Maintenance time has arrived.	The system-set regular maintenance time has arrived.	Please perform corresponding maintenance and then reset the maintenance time.
4307	The program is executed from a position other than the first line.	The program starts execution from a position other than the first line. You can press Cycle Start again to execute the program or reset to the first line to execute the program.	
4308	The running program is not the one being edited.	The running program is not the one being edited.	You can press Cycle Start again to execute the program, execute the currently edited program, or modify parameter No.3101#1.
4400	Measuring device not connected.	No measuring device detected or communication disconnected.	Please check the wiring or restart the measuring device.
4401	Tool offset data writing failed.	Please check if the system storage is normal or if the file is protected.	
4402	Exceeding the compensation range of the device.	The measured data exceeds the maximum compensation range specified by the device.	Please check if the measured value is normal or modify the compensation range.

Alarm No.	Significance	Possible Causes	Handling Method
4403	Exceeding maximum tool offset.	The specified tool offset to be written exceeds the maximum value.	Please check the tool offset number set for compensation.
4404	Specified device out of range.	The specified measuring device exceeds the maximum load capacity.	
4405	Failed to obtain measurement data.	The specified device position exceeds the detected quantity.	Please modify the specified position or check the wiring.
4410	Service life exhausted, system has stopped.	Please replace the tool whose service life has been exhausted and reset the used life of the exhausted tool to zero.	Press reset to clear this alarm.
4600	In spindle synchronization control based on servo motor, the spindle is not set as Cs axis.	Both the spindle with this function enabled and its master axis must be set as Cs axes.	Please check system parameters No.3800#7 and No.3701#7.
4601	In spindle synchronization control based on servo motor, the spindle does not enable this function but sets the relevant master axis number.	When a spindle is set with a master axis number, both the spindle and its master axis must have this function enabled.	Please check system parameters No.3800#7 and No.3810.
4602	In spindle synchronization control based on servo motor, setting a cascaded master axis is not allowed.	When a spindle is used as the master axis of another axis, its parameter No.3810 must be set to 0.	Please check system parameter No.3810.
4603	In spindle synchronization control based on servo motor, the set master axis number exceeds the maximum number of spindles.		Please check system parameters No.3710 and No.3810.
4604	In spindle synchronization control based on servo motor, the slave axis working mode cannot be specified.	The slave axis cannot find the specified master axis.	Please check system parameters No.3801 and No.3810.

Alarm No.	Significance	Possible Causes	Handling Method
5000	Physical connection disconnected.	Communication failure caused by chain breakage or interference.	Please power on again.
5005	Communication configuration failed.		Please power on again.
5006	The number of slave stations set in system parameters does not match the actual number.		Please check the settings of system parameters No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, No.3060 as well as the address settings of each device station. After modifying and confirming they are correct, power on again.
5007	There is no corresponding logical address for the loop.		Please check the settings of system parameters No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, No.3060 as well as the address settings of each device station. After modifying and confirming they are correct, power on again.
5008	Master station communication returns CP0.		Please power on again.
5010	Bus initialization parameter setting error.		Please check the settings of system parameters No.1023, No.3717, No.3050, No.3051, No.3052, No.3053, No.3054, No.3060 as well as the address settings of each device station. After modifying and confirming they are correct, power on again.
5011	Ethernet communication disconnected.		Please check the communication connection, and power on again after confirming it is correct.

Alarm No.	Significance	Possible Causes	Handling Method
5020	MDT data lost.		Please check the working status of the device.
5021	MST data lost.		Please check the working status of the device.
5022	MDT data verification error.		Please check the working status of the device.
5023	GDT data verification error.		Please check the working status of the device.
5030	Device alarm.		Please check the working status of the device, and power on again after confirming it is correct.
5031	Device alarm.		Please check the working status of the device and press reset to cancel the alarm.
5040	Slave station stops communication.		Please check the working status of the device.
5041	The total length of periodic data exceeds the limit.		Please check the bus configuration.
5042	The periodic data length of some slave stations exceeds the limit.		Please check the bus configuration.
5043	Report MST loss and loop break.		Please check the communication line with the corresponding wire number.
5044	Bus communication alarm		Please check both interfaces of the slave station, the communication cables connected to both interfaces, and the devices connected to the two communication cables.
5045	Disconnected.		Please check this section of the communication cable and the devices connected to the communication cable.

Alarm No.	Significance	Possible Causes	Handling Method
5046	Bus module failed to locate the fault position.		Please check the first slave station with alarm No.100 and the previous device.
5100	IDN16, 24 configuration failed.		Please check whether the system parameters are set correctly and whether the device is in normal working condition.
5101	IDN32, 35 configuration failed.		Please check whether the system parameters are set correctly and whether the device is in normal working condition.
5102	IDN5030, 5031, 5033 configuration failed.		Please check whether the system parameters are set correctly and whether the device is in normal working condition.
5103	I/O unit configuration failed.		Please check the working status of the device.
5111	Enumeration timeout.	Communication failure caused by chain breakage or interference.	The system has attempted to automatically correct this error. Please decide to reset to cancel the alarm or power on again based on whether the current communication is normal.
5112	Ring B handshake failed.	Communication failure caused by chain breakage or interference.	The system has attempted to automatically correct this error. Please decide to reset to cancel the alarm or power on again based on whether the current communication is normal.
5113	Delay test failed.	Communication failure caused by chain breakage or interference.	The system has attempted to automatically correct this error. Please decide to reset to cancel the alarm or power on again based on whether the current communication is normal.

Alarm No.	Significance	Possible Causes	Handling Method
5114	Communication parameter configuration failed.	Communication failure caused by chain breakage or interference.	The system has attempted to automatically correct this error. Please decide to reset to cancel the alarm or power on again based on whether the current communication is normal.
5132	Device warning.	Possible causes include: 1. Device parameters have been modified; 2. The device has a warning.	Press reset to cancel the alarm.
5133			Please check the device's working status and press reset to cancel this alarm.
5198	Ethernet communication is being established.		Please wait for a moment before operation.
5199	Bus communication error.		
5200	Failed to read servo model, version and other information.		Please try to power on again.
5201	Failed to read servo information table.		Please try to power on again.
5210	Failed to read servo parameters.		Please try to power on again.
5211	Servo parameters have been imported.	Please select the servo parameters in the imported parameter file to take effect.	Running parameters cannot be saved before the imported parameters take effect.
5220	The servo parameters in the CNC servo parameter storage file are inconsistent with the read servo parameters.		Please select the effective servo parameters.
5400	Mapping table does not exist.	I/O unit mapping table file not found, and automatic creation of the mapping table file also failed.	Please press reset to clear this alarm.

Alarm No.	Significance	Possible Causes	Handling Method
5401	The system has not recorded model, version and other information.	The current model and version information has been automatically memorized.	Please press reset to clear this alarm.
5402	The mapping table detected incorrect address or parameter settings.	The error has been automatically corrected.	Press reset to clear this alarm.
5403	Failed to read model, version and other information.	Failed to read information from the remote device. Please check if the I/O unit is in normal working condition.	Please power on again.
5404	Inconsistent.	The model, version and other information of the I/O unit are inconsistent with those memorized in the mapping table, and the current configuration has been automatically written.	Press reset to clear this alarm.
5405	No mapping address for PLC or analog output has been set.	Please go to the I/O unit parameter page to check if it is necessary to set the PLC mapping address or analog output address.	Press reset to clear this alarm.
5406	Device configuration failed.	Failed to issue parameters to the remote device. Please check if the I/O unit is in normal working condition.	Please power on again.
5500	Failed to read information such as model and version.		Please power on again.
6000	The external workpiece origin offset exceeds the allowable value range.	The external workpiece origin offset exceeds the allowable value range.	Please modify the program.
6001	The tool offset value exceeds the allowable value range.	The tool offset value exceeds the allowable value range.	Please modify the program.
6002	System parameter value writing error.	Error in writing parameter number, parameter value or parameter type.	Please modify the program.
6005	The additional workpiece coordinate origin offset exceeds the allowable value range.	The additional workpiece coordinate origin offset exceeds the allowable value range.	Please modify the program.

Alarm No.	Significance	Possible Causes	Handling Method
6006	Failed to obtain the additional workpiece coordinate system offset value.	Failed to read the additional workpiece coordinate system offset value when updating the workpiece coordinate system.	Please check if the memory is normal.
6007	The workpiece coordinate system origin offset exceeds the allowable value range.	The workpiece coordinate system origin offset exceeds the allowable value range.	Please modify the program.
6010	Error in executing the tool life command.	The tool group is full, or the tool group does not exist, or the tool does not exist.	It is necessary to modify the program or tool life data.
6011	Error in outputting the tool change permission signal.	The machine tool is in the tool change prohibited area, and the tool change permission signal cannot be output.	Please modify the program.
6013	The G31 command did not detect the skip signal.	When executing the G31 command, no skip signal is detected.	Please check the program and ladder diagram.
6014	Incorrect setting of the G31 command skip signal.	When executing the G31 command, the set skip address X or G is incorrect.	Please check the relevant parameters No.3008, No.3012, No.3020, and No.3025.
6015	Failed to capture the automatic tool compensation arrival signal.	In the automatic tool compensation function (G36, G37), within the area specified by parameters, the measurement position arrival signal (XAE or EAE) is not turned on. There is a setting or operation error.	
6016	MST code completion signal output error.	The completion signal of the ladder diagram was detected as not reset when executing the MST code.	The ladder diagram needs to be modified.
6017	Conflict occurred during MT code execution.	A conflict occurred because a parallel program was running while executing the MT code.	The program or ladder diagram needs to be modified.
6018	Error in waiting for M code instruction.	The parallel program has an error in waiting for the M code instruction, missing the P value.	The program needs to be modified.
6019	Excessive fluctuation of spindle speed during thread machining.	The spindle speed did not reach or exceeded the specified speed during thread machining.	Modify the program or check parameter No.1469.

Alarm No.	Significance	Possible Causes	Handling Method
6020	Spindle speed too high during thread machining.	The specified spindle speed was too high during thread machining, causing the feed axis to not operate normally.	The program needs to be modified.
6021	Spindle speed too low or zero during thread machining.	The specified spindle speed was too low during thread machining, causing the feed axis to not operate normally.	The program needs to be modified.
6022	Spindle encoder one-turn signal not detected during thread machining.	The specified spindle speed was too low during thread machining, causing the feed axis to not operate normally.	The program needs to be modified.
6023	Error in thread pitch increment/decrement during thread machining.	The thread pitch decrement was too large during thread machining, causing the feed axis to not operate normally.	The program needs to be modified.
6024	Spindle encoder lines not in the range of 100~5000.	This type of spindle encoder is not supported for normal tapping.	Please check the parameter settings (No.3720 and No.3723) or replace the spindle encoder.
6025	Spindle rotation command signal (SFR, SRV) error during normal tapping.	Check whether the G signals SFR and SRV in the ladder diagram are correctly given or the encoder connection.	The program, parameters, or ladder diagram need to be modified or checked.
6026	Spindle speed too low or too high during normal tapping, causing the tapping axis to not feed normally.	Possible reasons: 1. The spindle speed S value commanded before tapping is zero or too large; 2. Abnormal feedback from the spindle encoder.	Modify the program or check the spindle encoder.
6027	Spindle encoder one-turn signal not detected during normal tapping.	Please check whether system parameter No.3723 has correctly set the channel number corresponding to the spindle encoder.	Please check whether the spindle encoder is faulty or connected properly.
6028	Timeout or abnormality in M code execution for spindle start/stop during normal tapping.	Please check whether the ladder diagram correctly handles the M codes for spindle forward/reverse rotation and stop.	Please check whether the M code is effective.
6029	Spindle encoder selection error.	Incorrect selection of the spindle encoder, making thread cutting or tapping impossible.	Please modify the ladder diagram or check parameter No.3723.

Alarm No.	Significance	Possible Causes	Handling Method
6030	Rigid tapping mode signal is off.	Possible reasons: 1. The RGTAP signal was not detected or the rigid tapping mode was not specified (via M29 or other M codes) before starting tapping; 2. The rigid tapping spindle selection signal was not sent correctly.	Modify the program or check the ladder diagram.
6031	Spindle not in position mode during rigid tapping mode.	When the CON signal (G27#7) is OFF, the program commands movement along the Cs contour control axis.	Modify the program or refer to the ladder diagram to find the reason why the signal is not ON.
6032	Spindle selection error in rigid tapping mode	Possible Reasons: 1. Please check whether the rigid tapping spindle selection signal in the ladder diagram is correct; 2. Please check whether the Cs axis parameter setting of the spindle is correct.	
6035	C-axis command error in spindle mode.	When the CON signal (G27#7) is OFF, the program commands movement along the Cs contour control axis.	Modify the program or refer to the ladder diagram to find the reason why the signal is not ON.
6036	Spindle not in position mode during G28 command.	When the CON signal (G27#7) is OFF, the program commands movement along the Cs contour control axis.	Modify the program or refer to the ladder diagram to find the reason why the signal is not ON.
6037	Spindle speed arrival signal not detected.	During cutting, the spindle speed arrival signal (SAR) was not detected as valid.	Modify the program or check the ladder diagram.
6038	Waiting for rigid tapping signal RGTAP to be released.	Error in RGTAP signal when releasing rigid tapping.	Please modify the ladder diagram or check parameter No.5200#2.
6039	Stored data error during rigid tapping return.	The data stored during rigid tapping is incorrect, making rigid tapping return impossible.	Please re-store the data.
6040	Workpiece axis specification error in polygon machining.	In polygon machining, the workpiece axis does not specify a rotation axis or the Cs axis is set incorrectly.	Please modify the program or parameters.
6041	Illegal command of synchronization mode in polygon machining or helical tooth machining.	During synchronous operation in polygon machining or helical tooth machining, the NC program issued a movement command for the synchronous axis.	Please modify the program.

Alarm No.	Significance	Possible Causes	Handling Method
6042	Illegal command of synchronization mode in polygon machining.	During polygon machining, an attempt was made to perform both synchronous operation and CS contour control simultaneously.	Please modify the program.
6043	Workpiece axis not in position mode during polygon machining.	The workpiece axis did not enter position mode when polygon machining was specified.	Modify the program or refer to the ladder diagram to find the reason why the signal is not ON.
6044	Speed error of workpiece axis or tool axis in polygon cutting.	In polygon machining mode, the workpiece axis speed is too low or the tool axis speed exceeds the limit.	Please modify the program or parameter No.7621.
6045	Tool axis not in position mode during polygon machining.	The tool axis did not enter position mode when polygon machining was specified.	Modify the program or refer to the ladder diagram to find the reason why the signal is not ON.
6046	Encoder zero point not established for workpiece axis or tool axis in polygon machining.	In polygon machining mode, the encoder zero point for the workpiece axis or tool axis has not been established.	Modify the program or ladder diagram.
6047	Tool axis specification error in polygon machining.	In polygon machining, the tool axis does not specify a rotation axis or the Cs axis is set incorrectly.	Please modify the program or parameter No.7610.
6048	Conflict in setting of workpiece axis and tool axis in polygon machining.	In polygon machining, the same axis number was specified for both the workpiece axis and tool axis.	Please modify the program, spindle selection signal in the ladder diagram, or parameter No.7610.
6049	Spindle encoder one-turn signal not detected during polygon machining.	The encoder PC signal was not detected during polygon machining, causing the feed axis to not operate normally.	Modify the program or check the encoder.
6050	Illegal variable number used in macro program.	In the user macro program, a value that cannot be specified was designated as a variable number.	The program needs to be modified.
6051	Macro variable write protection.	This macro variable has been set to prohibit writing.	Please check parameters No.6031 and No.6032.
6052	Macro variable write prohibition.	This macro variable is read-only and cannot be written to.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6053	Writing to system macro variables cannot be empty.	When assigning values to system macro variables, empty values cannot be assigned.	The program needs to be modified.
6054	Macro variable data exceeds the allowed value range.	The set value of the macro variable exceeds the allowed value range.	The data needs to be modified.
6060	Competition between NC and PLC axis control commands.	NC commands and PLC axis control commands compete with each other.	Please modify the program or ladder diagram.
6061	Cannot change PLC-controlled axis.	PLC axis selection was performed for an axis under PLC axis control.	Please modify the ladder diagram.
6067	PLC signal prohibits operation.	A PLC signal was detected that prohibits the movement of the axis in the program.	Please modify the program or signal G261.
6068	Tool life has expired, and the system has stopped.	Please replace the tool with expired life and reset the used life of the expired tool to zero.	Press the reset button to clear this alarm.
6069	Machined part count reached alarm.	The number of machined parts has reached the set required number of parts. Please modify the number of machined parts.	
6070	Unsuccessful PC signal detection leading to spindle encoder value error.	Failure to correctly obtain the encoder reading due to unconnected drive unit, disconnected bus, or spindle not rotating in place.	Please check the bus connection or rotate the spindle.
6075	Tilted axis has not completed reference point return.	In the automatic reference point return state where manual reference point return in tilted axis control and no reference point return operation has been performed after power-on, an attempt was made to perform the reference point return operation of the orthogonal axis without completing the reference point return of the tilted axis.	The reference point return operation of the orthogonal axis must be performed after completing the reference point return operation of the tilted axis.
6081	Multiple spindle control commands in handwheel retraction program segment.	If multi-spindle control is used during handwheel retraction, the spindle speed will be incorrect.	Please modify the program.

Alarm No.	Significance	Possible Causes	Handling Method
6084	Stored travel detection alarm.	The command pre-judgment before program operation exceeds the stored travel detection range.	Please modify the program.
6085	Command error when servo enable is disconnected.		Please check the equipment working status.
6089	Waiting M code mismatch.	The main program and the additional axis program specified different waiting M codes.	Please modify the program.
6090	Servo working mode error in torque control.	For example, a torque command was received in position mode or a position command was received in torque mode.	Press the [Reset] key to cancel the alarm.
6091	Command torque value exceeds range.	The command torque value exceeds the value set in servo parameter No.177.	Please modify the program or servo parameters.
6092	F or S value under command torque exceeds range.	The F or S value under the command torque exceeds the value set in servo parameter No.83.	Please modify the program or servo parameters.
6093	Detection error occurred in torque command.	During the operation of the torque command, the torque arrival signal was not detected within the set stroke. Please modify the program.	
6094	Detection error occurred in torque command.	During the operation of the torque command, a torque limit signal was detected.	Please check the machine tool signal.
6095	Illegal synchronization mode command.	In the feed axis synchronization control, an abnormality occurred during synchronous operation, and the program issued a movement command to the slave axis.	Please modify the program.
6096	Illegal command in servo spindle synchronization control.	In the servo spindle synchronization control, the program commanded the movement of the driven axis.	Please modify the program.
6098	Operation conflict occurred in torque command.	During the operation of the torque command, it was detected that a parallel program was running the torque command simultaneously.	Please modify the program.

Alarm No.	Significance	Possible Causes	Handling Method
6099	Direction error occurred in torque command.	During the operation of the torque command, the set detection direction is inconsistent with the actual operation direction.	Please modify the program.
6100	Axis specification error in electronic gearbox function.	In the electronic gearbox function, the workpiece axis and tool axis specified by parameter No.7710 are incorrect.	The parameters need to be modified.
6101	Axial axis specification error in electronic gearbox function.	In the electronic gearbox function, the axial axis specified by parameter No.7711 is incorrect.	The parameters need to be modified.
6102	Tool axis working mode error in electronic gearbox function.	In the electronic gearbox function, the tool axis is not in position mode.	The ladder diagram needs to be modified.
6103	Workpiece axis working mode error in electronic gearbox function.	In the electronic gearbox function, the workpiece axis is not in position mode.	The ladder diagram needs to be modified.
6104	Axis attribute specification error in electronic gearbox function.	In the electronic gearbox function, the attributes of the workpiece axis and tool axis specified by parameter No.7710 are incorrect.	The parameters need to be modified.
6105	Speed specification error in electronic gearbox function.	In the electronic gearbox function, the specified speed of the tool axis exceeds the value set in parameter No.7721.	Please modify the program or parameters.
6106	Encoder zero point not established for workpiece axis or tool axis in electronic gearbox function.	In the electronic gearbox function, the encoder zero point for the workpiece axis or tool axis has not been established.	Modify the program or ladder diagram.
6110	Rotating axis working mode error in oil groove command.	In the oil groove command, the rotating axis is not in position mode.	The ladder diagram needs to be modified.
6111	Incorrect spindle setting for graphite wheel in G211 command.	In the G211 command, the spindle setting corresponding to the graphite wheel is incorrect.	Please modify the program or parameters.

Alarm No.	Significance	Possible Causes	Handling Method
6112	Incorrect following spindle setting for graphite wheel in G211 command.	In the G211 command, the setting of the spindle that the graphite wheel follows is incorrect.	Please modify the program or parameters.
6113	Incorrect graphite wheel diameter setting in G211 command.	In the G211 command, the diameter setting of the graphite wheel is incorrect.	Please modify the parameters.
6114	M command execution error in G211 command.	In the G211 command, an error occurred in the execution of the M command.	The ladder diagram needs to be modified.
6115	Indexing table working signal error.	During the indexing table movement command, the waiting for the release or clamping completion signal timed out.	The ladder diagram needs to be modified.
6119	Manual intervention and return error.	The number of segments moved during manual intervention and return exceeds the limit.	Reset to release.
6120	Excessive speed during rigid thread machining.	In rigid thread machining, the feed axis speed exceeds the value set in parameter 1422.	Please modify the program.
6126	Slave axis command error under flexible synchronization control.	Under flexible synchronization control, the movement amount of the slave axis was commanded.	Please modify the program.
6200	Fixed cycle command issued in non-XZ plane.	Fixed cycle commands must be issued in the XZ basic coordinate system.	The program needs to be modified.
6201	Axis outside ZX basic coordinate system specified in cycle.	An axis address outside the ZX plane was commanded in the fixed cycle.	The program needs to be modified.
6202	Absolute value of R greater than value of U (radius value) in G90/G77, G92/G78 commands.	When using G90/G92 commands (G77/G78 for B-set codes), if the signs of R and U are inconsistent, the absolute value of R is greater than the absolute value of U (radius value).	The program needs to be modified.
6203	Absolute value of R greater than value of W in G94/G79 command.	When using the G94 command (G79 for B-set codes), if the signs of R and W are inconsistent, the absolute value of R is greater than the absolute value of W.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6210	Illegal plane selection in multiple cycles.	When commanding multiple cycles G70~G76, the current plane is not XZ (modal G18), or plane switching commands G17~G18 are commanded in the G70~G76 program block.	The program needs to be modified.
6211	Axis address outside ZX plane commanded in G70~G76.	An axis address outside the ZX plane was commanded in the G70~G76 command or within the cycle body.	The program needs to be modified.
6212	Incorrect G code in G70~G73.	In G70~G73, an unusable G code was commanded between the two program blocks specified by addresses P and Q. Only G0~G3, G96/G97, G98/G99, and G40~G42 can be commanded in the NS~NF blocks.	The program needs to be modified.
6213	G70~G73 commands cannot run in MDI mode.	G70~G73 with P and Q were commanded in MDI (Manual Data Input) mode.	The program needs to be modified.
6214	Macro statements not allowed in G70~G73 cycles.	Execution of macro statements is not permitted in G70~G73 cycles.	The program needs to be modified.
6215	Subprogram called in G70~G73 cycles.	Subprogram calls are not allowed in G70~G73 cycles.	The program needs to be modified.
6216	Subprogram called in G70~G73 command line.	Subprogram calls are not allowed in the G70~G73 command line.	The program needs to be modified.
6217	Error in address P command in G70~G73 commands.	In G70~G73, P is not specified or the commanded value is out of range.	The program needs to be modified.
6218	Error in address Q command in G70~G73 commands.	In G70~G73, Q is not specified or the commanded value is out of range.	The program needs to be modified.
6219	Address P or Q not found in G70~G73 commands.	In the G70, G71, G72, or G73 command, the sequence number specified by address P or Q was not retrieved.	The program needs to be modified.
6220	Same commanded values for P and Q in G70~G73 commands.	The commanded values of P and Q are the same in the G70~G73 commands.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6221	Two consecutive G71~G73 command blocks not found in multiple cycle commands.	The two command blocks of the multiple cycle commands G71~G73 must be commanded consecutively; otherwise, errors may occur.	The program needs to be modified.
6222	Ns-Nf program blocks exceed 200 blocks in G70~G73 commands.	The number of Ns-Nf program blocks in the G70~G73 commands is too large, exceeding the maximum allowed 200 blocks.	The program needs to be modified.
6223	Tool feed direction same as path direction in Ns-Nf blocks of G71~G73 commands.	The tool feed direction is the same as the machining path direction, making it impossible to form a closed path.	The program needs to be modified.
6224	Tool feed direction same as finish turning allowance direction in Ns-Nf blocks of G71~G73 commands.	The tool feed direction is the same as the finish turning allowance direction in the G71~G73 cycle commands.	The program needs to be modified.
6225	Non-monotonic arc in Ns-Nf blocks of G71~G73 commands.	The arc in the Ns-Nf program blocks of G70~G73 is a major arc.	The program needs to be modified.
6226	Non-monotonic X-axis for arc in Ns-Nf blocks of G71~G72 commands.	The X-axis trajectory of the arc in the Ns-Nf program blocks of G71 Type I or G72 is non-monotonic.	The program needs to be modified.
6227	Non-monotonic Z-axis for arc in Ns-Nf blocks of G71~G72 commands.	The Z-axis trajectory of the arc in the Ns-Nf program blocks of G71 Type I or G72 is non-monotonic.	The program needs to be modified.
6228	Non-monotonic X-axis for arc in Ns-Nf blocks of G71 Type II commands.	The X-axis trajectory of the arc in the Ns-Nf program blocks of G71 Type II is non-monotonic.	The program needs to be modified.
6229	Non-monotonic X-axis for arc in Ns-Nf blocks of G73 commands.	The X-axis trajectory of the arc in the Ns-Nf program blocks of the G73 command is non-monotonic.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6230	Non-monotonic Z-axis for arc with finish turning allowance in Ns-Nf blocks of G73 commands.	In the Ns-Nf program blocks of G73, when the finish turning allowance is not zero, the Z-axis trajectory of the arc is non-monotonic.	The program needs to be modified.
6231	Non-monotonic Z-axis for arc with retract amount in Ns-Nf blocks of G73 commands.	In the Ns-Nf program blocks of G73, when the retract amount is not zero, the Z-axis trajectory of the arc is non-monotonic.	The program needs to be modified.
6232	Non-monotonic Z-axis for arc in Ns-Nf blocks of G72 Type II commands.	The Z-axis trajectory of the arc in the Ns-Nf program blocks of G72 Type II is non-monotonic.	The program needs to be modified.
6233	X-axis positioning point of G71/G72 commands within cutting range.	The X-axis positioning point of the G71/G72 commands is within the cutting range, which may cause tool collision. Please check parameter No.5104#2 FCK.	The program needs to be modified.
6234	Z-axis positioning point of G71/G72 commands within cutting range.	The Z-axis positioning point of the G71/G72 commands is within the cutting range, which may cause tool collision. Please check parameter No.5104#2 FCK.	The program needs to be modified.
6235	X-axis positioning point of G73 command within cutting range.	The X-axis positioning point of the G73 command is within the cutting range, which may cause tool collision. Please check parameter No.5104#2 FCK.	The program needs to be modified.
6236	Z-axis positioning point of G73 command within cutting range.	The Z-axis positioning point of the G73 command is within the cutting range, which may cause tool collision. Please check parameter No.5104#2 FCK.	The program needs to be modified.
6237	Excessive number of grooves commanded in G71/G72 Type II machining trajectory.	The number of grooves commanded in G71 or G72 Type II machining exceeds 30.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6238	X-axis retract direction inconsistent with finish turning allowance direction in G73 cycle.	The X-axis retract direction commanded in the G73 cycle is opposite to the finish turning allowance direction.	The program needs to be modified.
6239	Z-axis retract direction inconsistent with finish turning allowance direction in G73 cycle.	The Z-axis retract direction commanded in the G73 cycle is opposite to the finish turning allowance direction.	The program needs to be modified.
6240	Excessive number of roughing divisions in G71/G72 Type II rough turning cycle.	The number of roughing divisions in G71/G72 Type II machining is too large.	The program needs to be modified.
6241	X-axis finish turning allowance U in G70~G73 commands out of range.	The X-axis finish turning allowance U of G70~G73 is out of the allowed range.	The program needs to be modified.
6242	Z-axis finish turning allowance W in G70~G73 commands out of range.	The Z-axis finish turning allowance W of G70~G73 is out of the allowed range.	The program needs to be modified.
6243	No G00 or G01 commanded in initial block of G71~G72 cycle.	G00 or G01 must be commanded in the initial block of the G71~G72 cycle.	The program needs to be modified.
6244	Commands other than G00~G03 in initial block of G73 cycle.	Commands other than G00~G03 were commanded in the initial block (NS block) of the G73 cycle.	The program needs to be modified.
6245	Calculated radius difference of arc in G71~G73 program blocks out of range.	The calculated radius difference of the arc in the G71~G73 program blocks is out of the allowed range.	The program needs to be modified.
6246	Error in X-axis command in initial block of G71 cycle.	The initial block (NS block) of the G71 cycle must specify an X-axis increment, and the increment cannot be zero.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6247	Error in Z-axis command in initial block of G72 cycle.	The initial block (NS block) of the G72 cycle must specify a Z-axis increment, and the increment cannot be zero.	The program needs to be modified.
6248	Single-cut feed rate out of range in G71 or G72 command.	The single-cut feed rate in the G71 or G72 command is not greater than 0 or exceeds the maximum feed rate.	The program needs to be modified.
6249	Single retract amount R(e) less than 0 in G71 or G72 command.	The single retract amount R(e) in the G71 or G72 command is less than 0.	The program needs to be modified.
6250	Total cutting amount out of allowed range in G73 command.	The total cutting amount in the G73 command is out of the allowed range.	The program needs to be modified.
6251	Cycle count R(d) out of allowed range in G73 command.	The cycle count R(d) in the G73 command is less than 1 or greater than 999 after truncating the decimal part.	The program needs to be modified.
6252	Feed direction same as rough turning allowance direction in G73 command.	The feed direction determined by the positioning point and NS block in the G73 command is the same as the rough turning allowance direction.	The program needs to be modified.
6253	Failed to write parameters related to G71~G76/G78 multiple cycles.	Failed to write parameters related to G71~G76/G78 multiple cycles. Please check if the parameter file is normal.	The program needs to be modified.
6254	Failed to read program lines in G70~G76/G78 cycles.	Failed to read the program lines in NS~NF of G70~G76/G78 during operation.	The program needs to be modified.
6255	Block skip code present and effective in G71~G73 command blocks.	In block skip mode, the multiple cycle commands G71~G73 do not have two consecutive command blocks (which is required), otherwise errors may occur.	The program needs to be modified.
6256	Grooves exist in Ns-Nf blocks of G71 or G72 Type I commands.	The X-axis (for G71) or Z-axis (for G72) in the cycle command is non-monotonic. Please add the Type II flag in the Ns block.	Modify the program or set parameter 5102#1 to 0.

Alarm No.	Significance	Possible Causes	Handling Method
6257	Non-monotonic machining trajectory in Ns-Nf blocks of G71 or G72 commands.	The Z-axis (for G71) or X-axis (for G72) in the cycle command is non-monotonic.	Please modify the program or set parameter 5102#1 to 0.
6260	Value of Q out of allowed range in G74/G75 commands.	The value of Q in the G74/G75 commands is out of the allowed range.	The program needs to be modified.
6261	Value of P out of allowed range in G74/G75 commands.	The value of P in the G74/G75 commands is out of the allowed range.	The program needs to be modified.
6262	Value of R(e) out of allowed range in G74 or G75 command.	The single retract amount R(e) in the G74 or G75 command is less than zero or exceeds the maximum retract amount.	The program needs to be modified.
6263	Value of R(Δd) out of allowed range in G74 or G75 command.	The retract amount R(Δd) when cutting to the end point in the G74 or G75 command is less than 0 or exceeds the maximum retract amount.	The program needs to be modified.
6270	X or Z-axis movement amount at thread end is 0 in G76/G78 commands.	The X or Z-axis movement amount at the thread end in the G76/G78 commands is 0.	The program needs to be modified.
6271	Thread finish turning times P(m) in G76/G78 cycle less than 1 or greater than 99.	The thread finish turning times P(m) in the G76/G78 cycle is less than 1 or greater than 99.	The program needs to be modified.
6272	Thread chamfer width P(r) in G76/G78 out of allowed range.	The thread chamfer width P(r) in G76/G78 is out of the allowed range.	The program needs to be modified.
6273	Tool tip angle P(a) in G76/G78 commands out of allowed range.	The tool tip angle P(a) in the G76/G78 commands is out of the allowed range.	The program needs to be modified.
6274	Q(Δd_{min}) out of range in G76/G78 commands.	The minimum infeed amount Q(Δd_{min}) in the G76/G78 commands is out of the allowed range.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6275	Finish machining allowance R(d) in G76/G78 out of allowed range.	The finish machining allowance R(d) in G76/G78 is less than one minimum increment or greater than the maximum allowed value.	The program needs to be modified.
6276	Thread taper R(i) in G76/G78 out of allowed range.	The thread taper R(i) in G76/G78 is out of the allowed range.	The program needs to be modified.
6277	Mismatch between R value and U value when machining tapered threads with G76/G78.	When machining tapered threads with G76/G78, the starting point is between the thread start point and thread end point.	The program needs to be modified.
6278	Thread height P(k) value not specified in G76/G78 commands.	The thread height P(k) value is not specified in the G76/G78 commands.	The program needs to be modified.
6279	Thread height in G76/G78 commands less than finish machining allowance.	The thread height in the G76/G78 commands is less than the finish machining allowance.	The program needs to be modified.
6280	Thread height in G76/G78 commands less than minimum cutting amount.	The thread height in the G76/G78 commands is less than the minimum cutting amount.	The program needs to be modified.
6281	Thread height in G76/G78 commands greater than distance between positioning point and thread end point.	The thread height in the G76/G78 commands is greater than the distance between the positioning point and the thread end point.	The program needs to be modified.
6282	Q value out of allowed range in G76/G78 commands.	The first cutting depth Q value is not specified or is out of range in the G76/G78 commands.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6283	Thread taper in G76/G78 commands greater than 45°.	The thread taper in the G76/G78 commands is greater than 45°.	The program needs to be modified.
6284	Thread taper in G76/G78 commands parallel to cutting edge.	The thread taper in the G76/G78 commands is parallel to the cutting edge, making cutting impossible.	The program needs to be modified.
6285	Incorrect thread taper or tool tip angle in G76/G78 commands.	Incorrect tool tip angle or thread taper is commanded in G76/G78, preventing normal cutting.	The program needs to be modified.
6286	Long axis of thread in G78 command is Z-axis.	The G78 command cannot machine face threads. Please check if the radius difference between the start and end points in the command is correct.	The program needs to be modified.
6287	Incorrect cutting edge entry method in G78 command.	The cutting edge entry method in the G78 command is incorrect. The command value range is 0~3.	The program needs to be modified.
6288	Incorrect infeed depth method in G78 command.	The infeed depth method in the G78 command is incorrect. The D command value range is 0~1.	The program needs to be modified.
6300	Illegal spindle speed command in normal tapping.	The S code is not specified or the spindle speed S value is 0 in normal tapping.	The program needs to be modified.
6301	No spindle speed S commanded in rigid tapping G84 or G88 program block.	In rigid tapping, when using G84/G88 to specify rigid tapping mode (parameter No.5200#0 set to 1), no S command is present in the G84/G88 program block.	The program needs to be modified.
6302	Spindle pull-out speed J command out of range in rigid tapping.	The spindle pull-out speed J value is out of range in rigid tapping.	Modify the program or parameter 3031.
6303	Illegal K command in tapping or drilling fixed cycles.	The repeat count K value specified in tapping or drilling fixed cycles is not within 1~99.	The program needs to be modified.
6304	Lead command F out of range in tapping.	The specified cutting feed rate or lead is out of range in tapping.	Please check the G98 and G99 modes.

Alarm No.	Significance	Possible Causes	Handling Method
6305	Imperial lead I command out of range in metric input for tapping.	When using imperial lead I with metric input in tapping, the value of the I command is out of the allowed range.	Please check the G98 and G99 modes.
6306	Incorrect program command in rigid tapping.	In rigid tapping, the M code for starting rigid tapping mode and the S value are not in the same block.	The program needs to be modified.
6307	Illegal movement axis command in rigid tapping mode.	In rigid tapping, a movement axis is specified between the M code for starting rigid tapping mode and G84.	The program needs to be modified.
6308	Invalid axis commanded in rigid tapping or drilling fixed cycles.	An invalid axis is specified in the G83~G89 drilling/tapping commands.	Please check the axis attributes set in parameter No.1022 or modify the program.
6309	Tapping axis changed in tapping or drilling axis changed in drilling fixed cycles.	The G84/G88 command is switched during tapping, or the G83/G87, G85/G89 commands are switched during drilling fixed cycles. For example: executing G87 in G83 mode, G88 in G84 mode, or G89 in G85 mode.	The program needs to be modified.
6310	Plane selection command commanded in rigid tapping mode.	Plane switching commands G17~G19 are commanded in rigid tapping mode.	The program needs to be modified.
6311	Tapping length too short in rigid tapping.	In rigid tapping G84/G88, the commanded tapping length (distance from R plane to hole bottom) is less than the lead.	The program needs to be modified.
6312	Each cutting amount Q of deep hole tapping less than retract amount.	In deep hole tapping (Q value commanded and Q not equal to 0), the commanded Q value is less than the retract amount set in parameter No.5213.	Modify the program or parameters.

Alarm No.	Significance	Possible Causes	Handling Method
6313	Duplicate data specification in tapping.	In rigid tapping, the M code or S code is specified again between the M code for starting rigid tapping mode and G84.	The program needs to be modified.
6314	Commanded M code not allowed in rigid tapping or drilling fixed cycles.	M codes that cannot be in the same block are commanded in the G83~G89 program block or in that mode. M codes that can be in the same block are as follows:1. For G83/G87 and G85/G89: C-axis clamping M code (set in parameter No.5110);2. For G84/G88: M code for specifying rigid tapping mode (set in parameter No.5210, e.g., M29 when parameter is 0) and C-axis clamping M code (set in parameter No.5110).	Modify the program or parameters.
6315	Incorrect setting of M code for locking C-axis in drilling fixed cycles.	The parameter #5110 setting for the M code that locks the C-axis in drilling fixed cycles is incorrect and cannot be set to M30.	Modify the program or parameters.
6316	Cannot command G84/G88 for tapping in G96 mode.	G84/G88 cannot be commanded for tapping in G96 mode.	The program needs to be modified.
6317	Incorrect command in rigid tapping or drilling fixed cycle mode.	In G83~G89 mode, cylindrical interpolation G7.1/G107, polar coordinate interpolation G12.1/G112, G13.1/G113, or polygon machining G51.2 commands are commanded.	The program needs to be modified.
6318	T code commanded in rigid tapping or drilling fixed cycles.	A T command is commanded in the G83~G89 command block.	The program needs to be modified.
6319	No hole bottom data commanded in rigid tapping or drilling fixed cycles.	No hole bottom data for the tapping axis or drilling axis is commanded in the G83~G89 command block.	The program needs to be modified.
6320	Tapping axis or drilling axis commanded repeatedly during tapping or drilling.	Both the basic axis and its parallel axis are commanded for the tapping axis or drilling axis during tapping or drilling.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6327	Excessive nesting levels of IF and WHILE in macro program.	The nesting level of conditional judgment statements IF or conditional loops WHILE in the program exceeds 5 levels.	Please modify the program.
6328	IF or WHILE not in one-to-one correspondence with END in macro program.	The conditional judgment statements IF or conditional loops WHILE do not correspond one-to-one with END statements in the program.	Please modify the program.
6329	Macro program jump command GOTO cannot jump into conditional statements.	The macro program jump command GOTO cannot jump into conditional judgment statements (If..else..end) and while do..end.	Please modify the program.
6330	Incorrect command used in macro program.	A function that cannot be used is commanded in the user macro program.	The program needs to be modified.
6331	Mismatched brackets used in macro program.	The brackets '[' and ']' used in the user macro program are mismatched.	
6332	Incorrect conditional operator commanded in macro program.	A non-existent conditional operator is used in the user macro program.	Please refer to the manual and modify the program.
6333	Format error in macro program.	There is an error in the format of <formula>.	The program needs to be modified.
6334	Illegal variable number used in macro program.	A value that cannot be specified is designated as a variable number in the user macro program.	The program needs to be modified.
6335	Macro program repeated call.	M98, G65, or G66 is called again in G66 mode in the same program.	The program needs to be modified.
6336	Bracket nesting error.	The number of bracket nestings exceeds the upper limit (5 levels).	The program needs to be modified.
6337	Illegal operation data.	The argument of SQRT is negative, the arguments of BCD and BIN are negative, or the value of the BIN argument cannot be converted to correct BCD code.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6338	Division by zero.	The divisor is specified as 0 (including $\tan 90^\circ$).	The program needs to be modified.
6339	Excessive multiple macro mode calls.	Macro calls and macro mode calls exceed 4 levels.	The program needs to be modified.
6340	Jump statements and other transfer macro commands cannot be used in DNC and MDI.	Jump statements and other transfer macro commands are used during DNC and MDI operations.	The program needs to be modified.
6341	Missing end statement.	DO-END does not correspond 1:1; or there are other illegal commands in the END block with incorrect format; or a transfer jumps into the loop area.	The program needs to be modified.
6342	Insufficient permission to execute macro variable assignment operation.	Insufficient permission to execute macro variable assignment operations in MDI or DNC.	The program needs to be modified.
6343	Illegal loop count.	In Don, $1 \leq n \leq 5$ is not satisfied.	The program needs to be modified.
6344	NC statements and macro call statements in the same program block.	NC statements and macro call program commands are mixed.	The program needs to be modified.
6345	Illegal macro sequence number.	The sequence numbers defined in transfer commands are not 1~99999 or cannot be retrieved.	The program needs to be modified.
6346	Illegal argument address.	An address not allowed in <argument> is commanded.	The program needs to be modified.
6347	Tool tip direction data error.	The tool tip direction data input using macro variables needs to be in the range of 0 to 9 after rounding.	The user macro program needs to be modified.
6348	Illegal variable value.	The variable value is incorrect or invalid.	The program needs to be modified.
6349	Data error in logical operation commands.	The data operated on by logical operation commands OR, XOR, AND are negative values.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6350	G67 mode call cancellation commanded.	G67 mode call cancellation is commanded without commanding G66 macro program mode call. Please check if it is necessary to write the G66 command. Refer to parameter No.6000#0 G67.	The program needs to be modified.
6351	Macro variable write protection.	This macro variable has been set to prohibit writing.	Please check parameters No.6031 and No.6032.
6352	Macro variable write prohibition.	This macro variable is read-only and cannot be written to.	The program needs to be modified.
6353	Floating-point overflow in macro variable operation.	The floating-point data exceeded the allowed range during macro variable operation ($\pm 1E47$ when parameter No.6008#0 is 1, $\pm 1E308$ when 0).	Please modify the program or parameter No.6008#0.
6354	Cannot call macro program using this M code.	This M code cannot be used to call macro programs.	The program needs to be modified.
6355	Writing to system macro variables cannot be empty.	When assigning values to system macro variables, empty values cannot be assigned.	The program needs to be modified.
6356	Cannot call macro program using this T code.	This T code cannot be used to call macro programs.	Please modify the program or parameter No.6008#5, parameters No.6060~No.6069.
6357	Other G commands commanded in G66/G67 program block.	G66/G67 commands cannot be in the same block as commands from other groups.	The program needs to be modified.
6358	Error in operator in condition of macro program IF.	Relational operators and logical operators are used simultaneously in the same level of operation in the user macro program.	Please refer to the manual and modify the program.
6359	N number for macro program jump command GOTO does not exist.	In the macro program jump command, the N number specified in the GOTO command does not exist.	The program needs to be modified.
6360	Format error in macro program Type A command.	The format of the macro program Type A command is incorrect, or commands other than HPQR are commanded.	Please check the command format for the function corresponding to H.

Alarm No.	Significance	Possible Causes	Handling Method
6361	Incorrect H command value in macro program Type A.	In the macro program Type A command, the value of the H command is out of range. Refer to the manual for detailed values.	The program needs to be modified.
6362	Incorrect H command format in macro program Type A.	In the macro program Type A command, the format of H is incorrect; it must be a pure number.	The program needs to be modified.
6363	Cannot command QR in macro program Type A.	For the function corresponding to the value of H in macro program Type A, P must be commanded, and Q and R cannot be commanded.	The program needs to be modified.
6364	Cannot command R in macro program Type A.	For the function corresponding to the value of H in macro program Type A, both P and Q must be commanded, and R cannot be commanded.	The program needs to be modified.
6365	Must command R in macro program Type A.	For the function corresponding to the value of H in macro program Type A, P, Q, and R must all be commanded.	The program needs to be modified.
6366	Incorrect R value in remainder function of macro program Type A.	In the remainder function, the divisor specified by R is 0 (including $\tan 90^\circ$).	The program needs to be modified.
6367	Incorrect macro program Type A command format.	When parameter No.6000#7 is set to 1, the H command must immediately follow G65 in macro program Type A. To use G65 macro calls, set parameter No.6000#7 to 0.	The program needs to be modified.
6368	Multiple macro program calls in the same block.	G65 or G66, or M/T/G macro program calls are commanded simultaneously in the same program line.	Please modify the program or parameter No.6008.
6370	Cannot determine intersection point in tool tip radius compensation mode.	The intersection point in tool tip radius compensation mode cannot be determined.	The program needs to be modified.
6371	Cannot establish or cancel tool tip radius compensation mode in arc commands.	Tool tip radius compensation mode was established or canceled during arc interpolation.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6372	Compensation plane switching not allowed in tool tip radius compensation mode.	The compensation plane was switched in tool tip radius compensation mode.	The program needs to be modified.
6373	Interference in arc/parabola program blocks in tool tip radius compensation mode.	In tool tip radius compensation mode, the start or end point of the arc/parabola coincides with the center, or the arc end point is not on the arc.	The program needs to be modified.
6374	Interference in G90 or G94 blocks in tool tip radius compensation mode.	Overcutting may occur when using tool tip radius compensation mode with G90 or G94.	The program needs to be modified.
6375	Overcut detected in interference check in tool tip radius compensation mode.	Overcutting may occur in tool tip radius compensation mode.	The program needs to be modified.
6376	Tool path direction differs from programmed path direction in tool tip radius compensation mode.	In tool tip radius compensation mode, the tool tip radius path direction differs from the programmed path direction (by 90° to 270°), which may cause overcutting.	The program needs to be modified.
6377	Execution of G41 or G42 not allowed in MDI mode.	G41 or G42 (tool tip radius compensation mode) was specified in MDI mode. Refer to parameter No.5008#4 MCR.	The program needs to be modified.
6378	Possible overcut when cutting inside a full circle.	When cutting inside a full circle with tool tip radius compensation, overcutting may occur. Refer to parameter No.5008#5 CNF.	The program needs to be modified.
6379	Possible undercut when machining steps smaller than tool radius.	When machining steps smaller than the tool radius with tool tip radius compensation, undercutting may occur. Refer to parameter 5008#6 CNS.	The program needs to be modified.
6380	Arc radius smaller than tool radius when machining inner circle.	When machining an inner circle with tool tip radius compensation, the arc radius is smaller than the tool radius, which may cause overcutting.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6381	Arc command encountered when temporarily canceling or establishing tool tip radius compensation mode.	In tool tip radius compensation mode, a G command requiring temporary compensation cancellation was commanded, resulting in arc commands during compensation establishment or cancellation.	The program needs to be modified.
6382	Arc radius difference exceeds range during tool compensation calculation.	The calculated radius difference of the arc command exceeds the range in tool tip radius compensation.	The program needs to be modified.
6383	Error detected in tool tip radius compensation mode.	Programming or operation error; error detected in tool tip radius compensation mode.	Please modify the program.
6384	Polar coordinate interpolation cannot be performed without tool offset execution.	When establishing polar coordinate interpolation, the tool offset for the linear axis in polar coordinates is not effective, preventing polar coordinate interpolation.	The program needs to be modified.
6385	Error in canceling cylindrical interpolation.	Cylindrical interpolation cannot be canceled in C tool compensation mode.	Please modify the program.
6386	Incorrect axis attribute settings for two axes in C tool compensation calculation.	When establishing tool compensation, the axis attribute settings for the two axes used in C tool compensation calculation based on plane information are incorrect.	Please modify the parameters.
6387	No intersection between two parabolas or between arc and parabola paths.	With the current tool tip radius, there is no intersection between two parabolas or between arc and parabola paths.	The program needs to be modified.
6388	Parabola curvature too small.	In tool tip radius compensation, the parabola curvature is too small (smaller than the tool radius), which may cause overcutting.	The program needs to be modified.
6389	Cannot calculate normal intersection point for parabolas.	In tool tip radius compensation, normal intersection points cannot be calculated for two parabolas.	The program needs to be modified.
6390	T command commanded in tool tip radius compensation mode.	Tool change command T cannot be commanded in tool tip radius compensation (G41/G42).	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6391	Excessive non-movement commands in C tool compensation state.	There are excessive non-movement commands between two movement blocks in tool tip radius compensation.	The program needs to be modified.
6392	Radius difference of arc or arc chamfer exceeds range.	The calculated radius difference of the arc or arc chamfer command exceeds the range.	The program needs to be modified.
6400	Chamfer function or graphic dimension programming function invalid.	The chamfer function or graphic dimension programming function is invalid.1. If the program contains ', R' or ', C', check if parameter No.8134#2 is set to 0;2. If the program contains ', A', check if parameter No.3453 is set to 0 or parameter No.8134 is set to 0.	The program needs to be modified.
6401	Direct drawing dimension input commands can only be used in automatic mode.	Direct drawing dimension input commands can only be used in automatic mode, not in MDI or DNC mode.	The program needs to be modified.
6402	Cannot command chamfer or corner R in the current program block.	Chamfer or corner R must be specified in program blocks with modal G01, G02/G03 commands.	The program needs to be modified.
6403	Not in G01, G02/G03 modal command after chamfer or corner R.	The program block following the block specifying chamfer or corner R is not a G01, G02/G03 command.	The program needs to be modified.
6404	Incorrect axis address after chamfer or corner R.	The movement axis in the program block following the chamfer or corner R block is not the axis specified by the plane selection.	The program needs to be modified.
6405	Next block after chamfer or corner R block is an end command.	In automatic or DNC mode, the block following the chamfer or corner R block is a program end block; or in MDI mode, the chamfer or corner R block is the last block.	The program needs to be modified.
6406	Chamfer or corner R block in NF segment of G71~G76.	The block specifying chamfer or corner R is the last block of the G71~G76 cycle segment.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6407	Plane selection command specified after chamfer or corner R.	A plane selection command is specified after the block with chamfer or corner R.	The program needs to be modified.
6408	Movement amount in chamfer or corner R block too small or calculated chamfer point not on path.	The axis movement amount in the chamfer or corner R block is smaller than the chamfer amount or corner R value, or the calculated chamfer point is not on the path.	The program needs to be modified.
6409	Incorrect data in chamfer or corner R.	Incorrect data is commanded in the chamfer or corner R.	The program needs to be modified.
6410	Excessive G04 dwell commands after chamfer or corner R.	Two or more G04 dwell commands are commanded in the block following the chamfer or corner R block.	The program needs to be modified.
6411	Non-movement command after chamfer or corner R.	An empty line or non-movement commands such as M/S/T/F are commanded in the block following the chamfer or corner R block.	The program needs to be modified.
6412	No end point or angle value in direct drawing dimension input command.	In direct drawing dimension input, the block following a block specifying only an angle (, A) must specify absolute coordinate command values and angle command values.	The program needs to be modified.
6413	Angle in direct drawing dimension input command too small to calculate end point.	In direct drawing dimension input, the commanded angle is less than 1°, making it impossible to correctly calculate the program block end point.	The program needs to be modified.
6414	', A' command in direct drawing dimension input is in the last block of the program.	In direct drawing dimension input, the ', A' command is in the last block of the program, making it impossible to correctly calculate the program block end point.	The program needs to be modified.
6415	', A' command in direct drawing dimension input is in the NF segment of G71~G76.	In direct drawing dimension input, the ', A' command is in the NF program block of multiple cycle commands G71~G76, making it impossible to correctly calculate the program block end point.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6416	End point not found.	In direct drawing dimension input, the end point of the program block cannot be found.	The program needs to be modified.
6417	Forbidden G code commanded in direct drawing dimension input.	A G code that cannot be specified in direct drawing dimension input is specified. The following G codes cannot be used: G codes in group 00 except G04; G codes in group 01 except G00, G01, G32; Group 07 G22/G23; Group 11 G17~G19 plane switching commands; and Group 08 G83~G89.	The program needs to be modified.
6418	Incorrect mode when specifying direct drawing dimension input.	The current G command mode does not allow commanding program blocks with direct drawing dimension input. Disallowed mode commands include: G7.1 cylindrical interpolation mode, G commands in group 01 except G00, G01, G32, and G83~G89 in group 08.	The program needs to be modified.
6419	Consecutive program blocks without movement commands commanded in direct drawing dimension input.	In a sequence of consecutive direct drawing dimension input commands, two or more program blocks with no movement are specified.	The program needs to be modified.
6420	Incorrect end point calculated from the angle commanded in direct drawing dimension input.	In direct drawing dimension input, the end point of one axis cannot be correctly calculated based on the commanded angle and the end point of the commanded axis.	The program needs to be modified.
6421	Incorrect intersection point calculated from two angles commanded in direct drawing dimension input.	In direct drawing dimension input, the intersection point calculated from only the angle command ', A' and the angle in the next block is incorrect.	The program needs to be modified.
6422	Incorrect command format for chamfer function or graphic dimension programming function.	The command format for the chamfer function or graphic dimension programming function is incorrect: 1. Parameter No.3405#4 is set to use the chamfer function or graphic dimension programming function without the comma ',' format; 2. If the program contains ',', check if parameter No.3405#4 is set to 0.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6430	Tool life group number exceeds allowed range.	The tool life group number is less than 1 or exceeds the maximum allowed value set in parameter No.6813. When parameter No.6813 is set to 0, the maximum number of groups is 128.	The program needs to be modified.
6431	Tool group number not found.	The tool commanded in the program is not set.	Modify the program or parameters.
6432	T code not found.	The T code command is not stored in the commanded tool life group.	The program needs to be modified.
6433	Illegal tool life data.	The set tool life value is too large or not set. When using time counting, check the time unit set in parameter No.6805#1.	Modify the set value.
6434	Mismatched tool life management commands.	In tool life management, T[]99 is not specified or incorrectly specified before commanding T[]88.	Reconfigure the settings.
6435	Batch wear compensation life exhausted.	The batch wear compensation life has been exhausted. Replace the tool, reset the data to zero on the compensation page, and then restart the program.	
6440	Specified sequence number or program block number not found.	When restarting the program, the specified sequence number or program block number cannot be found.	Please re-specify.
6441	Specified line for program restart cannot be in G71~G73 cycle program blocks.	The selected program restart line is the finish turning shape program block of the G71~G73 command, i.e., the NS~NF block.	Please re-specify the restart line.
6442	Restart line for program restart contains unallowable G code.	The selected program restart line contains a G code that does not support restart.	Please re-specify the restart line.
6443	Program restart cannot be performed in thread or rigid tapping commands.	The selected program restart line is a thread cutting command (G32, G33, G34), thread cutting cycle command (Group A G code G92 or Group B G code G78), compound fixed thread cutting cycle command (G76), or rigid tapping command (G84/G88).	Please re-specify the restart line.

Alarm No.	Significance	Possible Causes	Handling Method
6444	System variable operation detected during program restart search.	System variables cannot be operated during the program restart search.	Please re-specify.
6445	Disallowed code executed in MDI mode after program restart search is completed.	Only M, S, and T commands can be executed in MDI mode after the program restart search is completed.	The program needs to be modified.
6446	G28/G30 command detected during program restart search.	A G28/G30 command is detected during the search when the program start command is specified without performing a reference point return operation.	The program needs to be modified.
6450	Feed rate of R command in grinding fixed cycle out of range.	The feed rate R is not commanded in cutting feed or the commanded value of feed rate R is improper:1. When using Group A codes, the required R value differs between G98 and G99 modes; check the G98 and G99 modes;2. When using Group B codes, the required R value differs between G94 and G95 modes; check the G94 and G95 modes.	The program needs to be modified.
6451	IJ not commanded in grinding fixed cycle commands.	Neither I nor J is commanded in the grinding fixed cycle commands G171~G179.	The program needs to be modified.
6452	Commanded K out of range in grinding fixed cycle commands.	The total cutting length K is not commanded or the commanded K is out of range in the grinding fixed cycle commands G175~G179.	The program needs to be modified.
6453	Commanded L out of range in grinding fixed cycle commands.	The repeat count L is out of range in the grinding fixed cycle commands G175~G179.	The program needs to be modified.
6454	Axis commands other than Z commanded in grinding fixed cycle commands G171~G174.	Axis commands other than the Z-axis are commanded in the grinding fixed cycle commands G171~G174.	The program needs to be modified.
6455	Axis commands other than XZ commanded in grinding fixed cycle commands G175, G177.	Axis commands other than the X and Z axes are commanded in the grinding fixed cycle commands G175, G177.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6456	Both X and Z axis commands commanded in grinding fixed cycle commands G175, G177.	Both X-axis and Z-axis commands are commanded in the grinding fixed cycle commands G175, G177.	The program needs to be modified.
6457	Axis commands other than X commanded in grinding fixed cycle commands G178, G179.	Axis commands other than the X-axis are commanded in the grinding fixed cycle commands G178, G179.	The program needs to be modified.
6458	Incorrect axis settings for I and J in grinding fixed cycle commands G171, G179.	The axes corresponding to I and J in the grinding fixed cycle commands G171, G179 do not exist in parameter 1020.	Modify the parameters or the program.
6470	Helical tooth machining command G51.3 commanded in C tool compensation mode.	The polygon machining command G51.3 is specified in C tool compensation mode (non-G40 mode).	The program needs to be modified.
6471	Incorrect workpiece tooth number TL command in helical tooth command G51.3.	T or L is missing in the command block of the helical tooth function G51.3, or the commanded value of the workpiece tooth number T/L is out of range.	The program needs to be modified.
6472	Incorrect P command in helical tooth command G51.3.	P is not specified in the G51.3 program block, or the specified P value is out of range.	Please modify the program.
6473	Incorrect Q command in helical tooth command G51.3.	Q is not specified in the G51.3 program block, or the specified Q value is out of range.	Please modify the program.
6474	Incorrect command for helical tooth command G51.3.	In helical tooth mode, commands such as G7.1, G12.1, and G51.2 cannot be commanded.	Please modify the program.
6475	Incorrect tool axis speed command S in helical tooth command G51.3.	The tool axis speed S command value is less than 0 in the G51.3 program block.	Please modify the program.
6490	Unallowable commands commanded in chip breaking command G7.9 program block.	G commands from other groups, MST commands, or axis commands are commanded in the program block of the chip breaking interpolation command G7.9.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6491	Incorrect P value commanded in chip breaking command G7.9 program block.	When K=0, P>0; when K=1, 1≤P≤4.	The program needs to be modified.
6492	Incorrect Q value commanded in chip breaking command G7.9 program block.	When K=0, Q>0; when K=1, 0<Q<99.999 (for ISB) or 0<Q<9.9999 (for ISC).	The program needs to be modified.
6493	Incorrect K value commanded in chip breaking command G7.9 program block.	In the program block of the chip breaking interpolation command G7.9, K cannot be omitted, and the K value can only be 1 or 0.	The program needs to be modified.
6494	Incorrect P command value for minor axis retract direction of G32 thread in chip breaking command G7.9 mode.	In chip breaking mode, the P command value for the minor axis retract direction of the G32 thread command is incorrect; P can only be commanded as 1 or 2.	The program needs to be modified.
6495	Incorrect minor axis retract direction of G32 thread command in chip breaking command G7.9 mode.	In chip breaking mode, the direction of the minor axis retract J value of the G32 thread command is inconsistent with the direction commanded by P.	The program needs to be modified.
6500	Axis number of indirect axis command out of range.	The axis number of the indirect axis command exceeds the maximum number of axes (maximum controlled axes: P1010; maximum spindle controlled axes: P3710).	The program needs to be modified.
6501	Axis name read by macro program AXNUM[] does not exist.	The axis name commanded via the macro program AXNUM[] does not exist. Check the correctness of the axis name and axis name subscript.	The program needs to be modified.
6502	Incorrect format for parameter reading via macro program PRM.	The format for reading parameters via the macro program PRM is incorrect. The correct format is (PRM[,I]) or (PRM[,/I]).	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6503	Byte command for parameter reading via macro program PRM out of range.	When reading parameters via the macro program PRM, the commanded number of bytes is out of range (0~7).	The program needs to be modified.
6504	Commanded parameter number for parameter reading via macro program PRM does not exist.	When reading parameters via the macro program PRM, the commanded parameter number does not exist.	The program needs to be modified.
6505	Macro variable in indirect axis command is empty.	The macro variable commanded in the indirect axis command is empty.	The program needs to be modified.
6506	Failed to read parameters via macro program PRM.	When reading parameters via the macro program PRM, the read parameter is an axis-type parameter, but no axis number is commanded.	The program needs to be modified.
6507	No data following the axis in the indirect axis command.	No assignment using the '=' sign is provided following the axis in the indirect axis command.	The program needs to be modified.
6600	Incorrect commanded movement value for CS axis.	The commanded movement value of the CS axis is not an integer multiple of the set value of the corresponding axis in parameter No.3470.	The program needs to be modified.
6610	Incorrect P value in G200/G201 commands.	The commanded P value in the program cannot exceed two digits.	The program needs to be modified.
6611	Incorrect (t) value in P(t)(s) value of G200/G201 commands.	The t value in the program can only be commanded as 1 or 2.	The program needs to be modified.
6612	Incorrect (s) value in P(t)(s) value of G200/G201 commands.	The s value in the program can only be commanded as 1 or 0.	The program needs to be modified.
6613	Incorrect commanded length Z(W) of closed groove in G200/G201 commands.	The length Z(W) of the closed groove is not specified in the program, or the commanded groove length is 0.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
6614	Incorrect commanded starting angle value C in G200 command.	For the G200 command, incremental programming cannot be used for the C-axis, and the C value can only be 0, 90, or 180.	The program needs to be modified.
6615	Incorrect rotary axis C in G200/G201 commands.	The C-axis is not commanded in the program, or the commanded C-axis is not a rotary axis.	The program needs to be modified.
6616	Incorrect linear axis Z in G200/G201 commands.	The Z-axis is not commanded in the program, the Z-axis is commanded with absolute instructions, or the commanded Z-axis is not a linear axis.	The program needs to be modified.
6617	Incorrect feed axis X in G200/G201 commands.	The X-axis is not commanded in the program, or the commanded X-axis is not a linear axis.	Modify the program or parameters.
6620	Incorrect interpolation axis parameter settings for G211 command.	The parameters No.5330 and No.5331 (which set the lifting table axes for the G211 command) are set to rotary axes or invalid axes.	Modify parameters No.5330, No.5331, or No.1006#0 and #1.
6621	Axes other than the lifting table control axes commanded in G211 command.	Axes other than those set in parameters No.5330 or No.5331 are commanded in the G211 program.	Modify the program or parameters.
6622	Two interpolation axes of G211 command must be commanded simultaneously.	In the G211 program block, the two interpolation axes must be commanded at the same time.	The program needs to be modified.
6623	Incorrect value of division count L in G211 command.	In the G211 program block, the value of the division count L cannot be 0.	The program needs to be modified.
6700	G221 or G220 not specified in a single block.		Please modify the program.
6701	G221 repeatedly specified after enabling follow-up function with G221.		Please modify the program.

Alarm No.	Significance	Possible Causes	Handling Method
6702	P value not specified or the program specified by P value does not exist when commanding G221.		Please modify the program.
6703	K value not specified or out of range (0~100) when commanding G221.		Please modify the program.
6704	F value not specified or out of range (0~500) when commanding G221.		Please modify the program.
6705	Missing G220 command to end the program after enabling follow-up function with G221.		Please modify the program.
6900	Error in parallel program pre-reading command.	The positions of G100 and G101/G102/G103/G104 commands specified in the main program are incorrect, or G101/G102/G103/G104 are repeatedly specified.	The program needs to be modified.
6901	Parallel program pre-reading command not specified in a single block.	Parallel program pre-reading commands must be specified in a single block.	The program needs to be modified.
6902	Parallel program pre-reading program blocks exceed range.	The number of program blocks for parallel program pre-reading exceeds the 100-line range.	The program needs to be modified.
7001	Failed to open auxiliary axis part program.	Press the reset key to clear the alarm, then reselect and execute the program.	
7002	Failed to read auxiliary axis program line.	Failed to read program line code during MDI program operation or syntax check.	Press the reset key to clear the alarm.
7009	Operator format error in auxiliary axis program.	The format of the operation command in the auxiliary axis program block is incorrect.	Please modify the program.

Alarm No.	Significance	Possible Causes	Handling Method
7010	A program line in auxiliary axis program exceeds 256 characters.	The number of characters in one program line is too large.	Please modify the program.
7011	Data in auxiliary axis program exceeds allowed value range.	The input data exceeds the allowed value range, or the specified data exceeds 8 digits.	Modify the data.
7012	Address not found in auxiliary axis program.	There is no address at the beginning of the program block; only values or symbols are input.	The program needs to be modified.
7013	No data following the address in auxiliary axis program.	No data follows the address, or the format of the expression after the address is incorrect (parentheses not used).	The program needs to be modified.
7014	Incorrect use of minus sign in auxiliary axis program.	The minus sign "-" is used incorrectly (used where it is not allowed, or multiple minus signs are input).	The program needs to be modified.
7015	Incorrect use of decimal point in auxiliary axis program.	The decimal point "." is used incorrectly (used after commands that do not allow decimal points, or multiple decimal points are input).	The program needs to be modified.
7016	Illegal address input in auxiliary axis program.	An unavailable address is input in the valid information area.	The program needs to be modified.
7017	Incorrect G code in auxiliary axis program.	A G code that is not allowed to be commanded is used in the auxiliary axis program. Only G00/G01/G04/G28/G30 can be executed in the auxiliary axis program.	The program needs to be modified.
7018	Address duplication error in auxiliary axis program.	The same address is commanded twice or more in one program block, or two or more G codes from the same group are commanded in one program block. Refer to parameter No.3403#6 AD2.	The program needs to be modified.
7019	Record end in auxiliary axis program.	A record end character (%) is specified or no program end statement is specified. Refer to parameter No.3404#6.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
7040	Excessive M codes in auxiliary axis program.	More M codes than the number set in parameter No.3404#7 (or more than the maximum of 3) are commanded in the same program block.	Modify the program or parameters.
7041	M command value in auxiliary axis program out of parameter allowed range.	The number of digits of the M command exceeds the set value of parameter No.3030, or the value of the M code command is out of range. Modify the program or parameters.	The program needs to be modified.
7042	G code that cannot be in the same block as M99 commanded in auxiliary axis program.	Commands G28, G30, G53, or G36/G37 cannot be specified in the same program block as M99.	The program needs to be modified.
7075	Axis with invalid axis type setting commanded in auxiliary axis program.	The axis type setting of the commanded axis is invalid. Check the settings of parameters No.1006#0 and No.1006#1 for the axis.	Modify the program or parameters.
7076	Rotary axis that cannot execute interpolation commands commanded in auxiliary axis program.	A rotary axis is commanded in non-polar/cylindrical interpolation state and non-G00/G01 mode. Modify the program or check the settings of parameters No.1006#0 and No.1006#1.	The program needs to be modified.
7080	Axes other than basic axes commanded in auxiliary axis arc command.	The axes commanded in the arc command are not the basic axes set in parameter 1022.	Please modify the program or parameter No.1022.
7081	Rotary axis commanded in arc interpolation.	A rotary axis is commanded in auxiliary axis arc interpolation. Modify the program or check the settings of parameters No.1006#0 and No.1006#1.	Modify the program or parameters.
7082	No arc radius R or I/J/K commanded.	Neither R nor I, J, K is commanded in auxiliary axis arc interpolation.	Modify the program or parameter No.3403#5.
7083	Incorrect radius command value in arc interpolation.	In auxiliary axis arc interpolation, the end point is not on the arc specified by R. Refer to parameter No.3403#4.	The program needs to be modified.
7084	Arc commanded in auxiliary axis exceeds radius difference range.	In auxiliary axis arc interpolation, the difference between the distance from the start point to the center and the distance from the end point to the center exceeds the value set in parameter No.3410.	Modify the program or parameters.

Alarm No.	Significance	Possible Causes	Handling Method
7085	Illegal plane axis commanded in auxiliary axis arc interpolation.	An axis not in the selected plane is commanded in auxiliary axis arc interpolation.	The program needs to be modified.
7086	Auxiliary axis arc interpolation plane cannot be determined.	In auxiliary axis arc interpolation, the end positions of the two axes for interpolation must be commanded; otherwise, the interpolation plane cannot be determined. No axis command or only one axis command is present in the G02/G03 program block, making the arc interpolation plane undetermined.	The program needs to be modified.
7096	P or X of G04 command in auxiliary axis program out of range.	The dwell time P of the G04 command exceeds the range of 0~99999999, or X exceeds the range of -9999~9999.	The program needs to be modified.
7165	Reference point of commanded axis not established in auxiliary axis program.	The reference point of the commanded axis has not been established before cycle start.	Establish the reference point or modify parameter No.1005#0.
7166	G28 commanded before reference point established in auxiliary axis program.	The reference point has not been established before executing G28.	Modify the program or parameter No.1002#3.
7167	G30 commanded before reference point established in auxiliary axis program.	The reference point has not been established before executing G30.	
7168	Incorrect reference point P commanded in G30 of auxiliary axis program.	A value other than 2~4 is specified for P in the G30 command block.	The program needs to be modified.
7169	Intermediate point axis of G28/G30 command out of range in auxiliary axis program.	When returning to the reference point via G28/G30, the commanded intermediate point exceeds the allowed range, or the calculated machine coordinate intermediate point of the G28 command is out of range.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
7180	Codes that cannot be in the same block as G35 commanded in auxiliary axis program.	G35 commands cannot be in the same block as other G codes or M/T commands.	The program needs to be modified.
7181	Illegal torque command in G35 of auxiliary axis program.	No torque is commanded in the G35 command, or the commanded torque is less than 0.	Modify the program or parameters.
7182	Incorrect axis command in G35 of auxiliary axis program.	The torque command cannot be incremental, or two or more torque commands are commanded.	The program needs to be modified.
7183	F value commanded in G35 of auxiliary axis program out of range.	The F value commanded in the G35 command is out of the range (0.001~99999).	The program needs to be modified.
7184	No S and F commanded in G35 of auxiliary axis program.	Neither the S command nor the F command is commanded in the G35 command.	The program needs to be modified.
7185	Both J and K commanded in G35 of auxiliary axis program.	Alarm margin J and alarm flag K cannot be commanded simultaneously in the G35 command.	The program needs to be modified.
7186	Incorrect K value in G35 of auxiliary axis program.	The K value in the G35 command can only be 0 or 1.	The program needs to be modified.
7200	Chamfer function or graphic dimension programming function invalid in auxiliary axis program.	The chamfer function and graphic dimension programming function cannot be used in the auxiliary axis program.	The program needs to be modified.
7201	Block skip character cannot be used in auxiliary axis program.	The block skip function cannot be used in the auxiliary axis program.	The program needs to be modified.
7202	Macro statements cannot be used in auxiliary axis program.	Macro programs cannot be used in the auxiliary axis program.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
7203	Subprogram calls cannot be used in auxiliary axis program.	Subprogram call commands such as M98, M198, G65, and G66 cannot be used in the auxiliary axis program.	The program needs to be modified.
7330	Incorrect command used in macro program of auxiliary axis program.	A function that cannot be used is commanded in the user macro program.	The program needs to be modified.
7331	Mismatched brackets used in macro program of auxiliary axis program.	The brackets '[' and ']' used in the user macro program are mismatched.	The program needs to be modified.
7332	Incorrect conditional operator commanded in macro program of auxiliary axis program.	A non-existent conditional operator is used in the user macro program.	Please refer to the manual and modify the program.
7333	Format error in macro program of auxiliary axis program.	There is an error in the format of <formula>.	The program needs to be modified.
7334	Illegal variable number used in macro program of auxiliary axis program.	A value that cannot be specified is designated as a variable number in the user macro program.	The program needs to be modified.
7335	Macro program repeated call in auxiliary axis program.	M98, G65, or G66 is called again in G66 mode in the same program.	The program needs to be modified.
7336	Bracket nesting error in auxiliary axis program.	The number of bracket nestings exceeds the upper limit (5 levels).	The program needs to be modified.
7337	Illegal operation data in auxiliary axis program.	The argument of SORT is negative, the arguments of BCD and BIN are negative, or the value of the BIN argument cannot be converted to correct BCD code.	The program needs to be modified.
7338	Division by zero in auxiliary axis program.	The divisor is specified as 0 (including $\tan 90^\circ$).	The program needs to be modified.
7339	Excessive multiple macro mode calls in auxiliary axis program.	Macro calls and macro mode calls exceed 4 levels.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
7340	Jump statements and other transfer macro commands cannot be used in DNC and MDI of auxiliary axis program.	Jump statements and other transfer macro commands are used during DNC and MDI operations.	The program needs to be modified.
7341	Missing end statement in auxiliary axis program.	DO-END does not correspond 1:1;or there are other illegal commands in the END block with incorrect format;or a transfer jumps into the loop area.	The program needs to be modified.
7342	Insufficient permission to execute macro variable assignment operation in auxiliary axis program.	Insufficient permission to execute macro variable assignment operations in MDI or DNC.	The program needs to be modified.
7343	Illegal loop count in auxiliary axis program.	In Don, $1 \leq n \leq 3$ is not satisfied.	The program needs to be modified.
7344	NC statements and macro call statements in the same program block of auxiliary axis program.	NC statements and macro call program commands are mixed.	The program needs to be modified.
7345	Illegal macro sequence number in auxiliary axis program.	The sequence numbers defined in transfer commands are not 1~99999 or cannot be retrieved.	The program needs to be modified.
7346	Illegal argument address in auxiliary axis program.	An address not allowed in <argument> is commanded.	The program needs to be modified.
7347	Tool tip direction data error in auxiliary axis program.	The tool tip direction data input using macro variables needs to be in the range of 0 to 9 after rounding.	The user macro program needs to be modified.
7348	Illegal variable value in auxiliary axis program.	The variable value is incorrect or invalid.	The program needs to be modified.
7349	Data error in logical operation commands of auxiliary axis program.	The data operated on by logical operation commands OR, XOR, AND are negative values.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
7350	G67 mode call cancellation commanded in auxiliary axis program.	G67 mode call cancellation is commanded without commanding G66 macro program mode call. Please check if it is necessary to write the G66 command. Refer to parameter No.6000#0 G67.	The program needs to be modified.
7351	Macro variable write protection in auxiliary axis program.	This macro variable has been set to prohibit writing.	Please check parameters No.6031 and No.6032.
7352	Macro variable write prohibition in auxiliary axis program.	This macro variable is read-only and cannot be written to.	The program needs to be modified.
7353	Floating-point overflow in macro variable operation of auxiliary axis program.		
7354	Cannot call macro program using this M code in auxiliary axis program.	This M code cannot be used to call macro programs.	The program needs to be modified.
7355	Writing to system macro variables cannot be empty in auxiliary axis program.	When assigning values to system macro variables, empty values cannot be assigned.	The program needs to be modified.
7356	Cannot call macro program using this T code in auxiliary axis program.	This T code cannot be used to call macro programs.	Please modify the program or parameter No.6008#5, parameters No.6060~No.6069.
7357	Other G commands commanded in G66/G67 program block of auxiliary axis program.	G66/G67 commands cannot be in the same block as commands from other groups.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
7358	Excessive nesting levels of IF and WHILE in macro program of auxiliary axis program.	The nesting level of conditional judgment statements IF or conditional loops WHILE in the program exceeds 5 levels.	Please modify the program.
7359	IF/WHILE and END in auxiliary axis macro program do not correspond one-to-one.	The conditional judgment statements IF/conditional loops WHILE and END statements in the program do not correspond one-to-one.	Please modify the program.
7360	GOTO jump command in auxiliary axis macro program cannot jump into conditional statements.	The macro program jump command GOTO cannot jump into conditional judgment statements IF and WHILE.	Please modify the program.
7361	N number specified by GOTO jump command in auxiliary axis macro program does not exist.	In the macro program jump command, the N number specified by GOTO does not exist.	The program needs to be modified.
7400	Chamfer function or graphic dimension programming function invalid in auxiliary axis program.	The chamfer function or graphic dimension programming function is invalid. If the program contains ", R" or ", C", check if parameter No.8409#0 is set to 0; if the program contains ", A", check if parameter No.3453 is set to 0 or parameter No.8409 is set to 0.	The program needs to be modified.
7401	Direct drawing dimension input commands in auxiliary axis program can only be used in automatic mode.	Direct drawing dimension input commands can only be used in automatic mode, not in MDI or DNC mode.	The program needs to be modified.
7402	Chamfer or corner R cannot be commanded in the current program block of auxiliary axis program.	Chamfer or corner R must be specified in the program block commanded in G01/G02/G03 mode.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
7403	The mode after chamfer or corner R in auxiliary axis program is not G01/G02/G03.	The program block following the one specifying chamfer or corner R is not a G01/G02/G03 command.	The program needs to be modified.
7404	The axis address after chamfer or corner R in auxiliary axis program is not the required one.	The movement axis in the program block following the one specifying chamfer or corner R is not the axis specified by plane selection.	The program needs to be modified.
7405	The next block after the chamfer/corner R block in auxiliary axis program is an end command.	In automatic or DNC mode, the block following the one specifying chamfer or corner R is the program end block; in MDI mode, the block specifying chamfer or corner R is the last block.	The program needs to be modified.
7406	The chamfer/corner R block in auxiliary axis program is the NF block of G71~G76.	The program block specifying chamfer or corner R is the last block of the G71~G76 cycle.	The program needs to be modified.
7407	A plane selection command is commanded after the chamfer/corner R block in auxiliary axis program.	A plane selection command is specified after the program block specifying chamfer or corner R.	The program needs to be modified.
7408	The movement amount in the chamfer/corner R block of auxiliary axis program is too small, or the calculated chamfer point is not on the path.	The axis movement amount in the program block specifying chamfer or corner R is smaller than the chamfer amount or corner R value, or the calculated chamfer point is not on the path.	The program needs to be modified.
7409	Incorrect data in chamfer or corner R of auxiliary axis program.	Incorrect data is commanded for chamfer or corner R.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
7410	Excessive G04 dwell commands are commanded after the chamfer/corner R block in auxiliary axis program.	Two or more G04 dwell commands are commanded in the program block following the one specifying chamfer or corner R.	The program needs to be modified.
7411	Non-movement commands are commanded after the chamfer/corner R block in auxiliary axis program.	The program block following the one specifying chamfer or corner R contains a blank line or non-movement commands such as M/S/T/F.	The program needs to be modified.
7412	No end point or angle value for direct drawing dimension input command in auxiliary axis program.	In direct drawing dimension input, the program block following the one specifying only the angle (, A) must specify absolute coordinate command values and angle command values.	The program needs to be modified.
7413	The angle of direct drawing dimension input command in auxiliary axis program is too small to calculate the end point.	In direct drawing dimension input, the commanded angle is less than 1°, making it impossible to correctly calculate the end point of the program block.	The program needs to be modified.
7414	The direct drawing dimension input command ', A' in auxiliary axis program is in the last program block.	In direct drawing dimension input, the ', A' command is in the last program block of the program, making it impossible to correctly calculate the end point of the program block.	The program needs to be modified.
7415	The direct drawing dimension input command ', A' in auxiliary axis program is in the NF block of G71~G76.	In direct drawing dimension input, the ', A' command is in the NF program block of the multiple cycle commands G71~G76, making it impossible to correctly calculate the end point of the program block.	The program needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
7416	End point not found in auxiliary axis program.	In direct drawing dimension input, the end point of the program block cannot be found.	The program needs to be modified.
7417	Disallowed G codes are commanded for direct drawing dimension input in auxiliary axis program.	G codes that cannot be specified in direct drawing dimension input are commanded. The following G codes are not allowed:- G codes in group 00 except G04;- G codes in group 01 except G00 and G01.	The program needs to be modified.
7418	Incorrect mode when specifying direct drawing dimension input in auxiliary axis program.	The current G command mode does not allow commanding program blocks with direct drawing dimension input. Disallowed mode commands include: G7.1 cylindrical interpolation mode, G codes in group 01 except G00, G01, G32, and G83~G89 in group 08.	The program needs to be modified.
7419	Consecutive program blocks without movement commands commanded for direct drawing dimension input in auxiliary axis program.	In a sequence of consecutive direct drawing dimension input commands, two or more program blocks with no movement are specified.	The program needs to be modified.
7420	Incorrect end point calculated from the angle of direct drawing dimension input command in auxiliary axis program.	In direct drawing dimension input, the end point of one axis cannot be correctly calculated based on the commanded angle and the end point of the commanded axis.	The program needs to be modified.
7601		Failed to open part program.	
7602		Failed to read program line.	
7603		Failed to read program buffer.	
7609		Operator format error in program.	
7610		A program line exceeds 256 characters.	

Alarm No.	Significance	Possible Causes	Handling Method
7611		Data exceeds allowed value range.	
7612		Address not found.	
7613		No data following the address.	
7614		Incorrect use of minus sign.	
7615		Incorrect use of decimal point.	
7616		Illegal address input.	
7617		Incorrect G code.	
7618		Address duplication error.	
7619		Record end error.	
7620		Commanded axis address does not exist in program.	
7641		M command value out of parameter allowed range.	
7642		G code that cannot be in the same block as M99 is commanded.	
7680		Reference point of commanded axis not established.	
7681		G28 commanded before reference point established.	
7683		Incorrect reference point P commanded in G30.	
7685		P or X value of G04 command out of range.	
7686		Spindle selection P command out of range.	
7687		Spindle selection P_S_ command is commanded when multi-spindle parameter No.3710 is set to 1 or parameter No.3703#3=0.	
7688		Axis number n commanded is out of range when using Sn command for multi-spindle selection.	
7689		P command for multi-spindle selection not specified.	

Alarm No.	Significance	Possible Causes	Handling Method
7690		Codes that cannot be in the same block as G35 are commanded in G35.	
7692		Illegal torque command in G35.	
7693		Incorrect axis command in G35.	
7694		F value commanded in G35 out of range.	
7695		No S and F commanded in G35.	
7696		Both J and K commanded in G35.	
7697		Incorrect K value in G35.	
7704		The set value of default parameter 3775 or 3776 in multi-spindle selection function is greater than the total number of spindles.	
7705		P command value in G31 out of range.	
7706		Excessive axes commanded in G31 jump command.	
7707		Excessive nesting levels of IF and WHILE in macro program.	
7708		IF/WHILE and END in macro program do not correspond one-to-one.	
7709		GOTO jump command in macro program cannot jump into conditional statements.	
7710		Incorrect command used in macro program.	
7711		Mismatched brackets used in macro program.	
7712		Incorrect conditional operator commanded in macro program.	
7713		Format error in macro program.	
7714		Illegal variable number used in macro program.	
7715		Macro program repeated call.	
7716		Bracket nesting error.	
7717		Illegal operation data.	
7718		Division by zero error.	
7719		Excessive multiple macro mode calls.	
7720		Jump statements and other transfer macro commands cannot be used.	

Alarm No.	Significance	Possible Causes	Handling Method
7721		Missing end statement.	
7722		Insufficient permission to execute macro variable assignment operation.	
7723		Illegal loop count.	
7724		NC statements and macro call statements in the same program block.	
7725		Illegal macro sequence number.	
7726		Illegal argument address.	
7727		Tool tip direction data error.	
7728		Illegal variable value.	
7729		Data error in logical operation commands.	
7731		Macro variable write protection.	
7732		Macro variable write prohibition.	
7733		Floating-point overflow in macro variable operation.	
7734		Cannot call macro program using this M code.	
7735		Writing to system macro variables cannot be empty.	
7736		Cannot call macro program using this T code.	
7737		Other G commands commanded in G66/G67 program block.	
7739		N number specified by GOTO jump command in macro program does not exist.	
7750		Spindle speed S value error in rigid tapping.	
7751		No spindle speed S commanded in G84/G88 program block for rigid tapping.	
7752		K value out of range (1-99) in tapping or drilling fixed cycle.	
7753		Lead command F out of range in tapping. See the manual for details.	
7754		Inch lead I command out of range under metric input in tapping.	

Alarm No.	Significance	Possible Causes	Handling Method
7755		The block following the M29 program block is not a G84 program block.	
7756		Commanded axis invalid in rigid tapping or drilling fixed cycle.	
7757		Tapping axis changed in tapping or drilling axis changed in drilling fixed cycle.	
7758		Tapping length too short in rigid tapping.	
7759		T code commanded in rigid tapping or drilling fixed cycle.	
7760		No hole bottom data commanded in rigid tapping or drilling fixed cycle.	
7761		M and S codes for rigid tapping startup are not in the same block.	
7762		Tapping axis or drilling axis with the same axis attribute commanded during tapping or drilling.	
7763		M codes other than rigid tapping mode M codes or C-axis clamping M codes are commanded during rigid tapping or drilling.	
7764		The M code or S code is specified again between the rigid tapping mode M code and G84 for startup.	
7765		M code for locking C-axis in drilling fixed cycle is incorrectly set.	
7766		G84/G88 cannot be commanded for tapping in G96 mode.	
7767		Incorrect mode commanded in rigid tapping or drilling fixed cycle.	
7768		Spindle pull-out speed J command out of range in rigid tapping.	
7769		Plane selection command commanded in rigid tapping.	

Alarm No.	Significance	Possible Causes	Handling Method
7790		Excessive commands in the same block of parallel program.	
7791		Parallel program cannot call subprograms.	
7792		Parallel program command error.	
7800		Motion command conflicts with main program.	
7801		Motion command conflicts with each subprogram.	
7802		External alarm triggers program stop.	
7803		MST completion signal timeout error.	
7804		G35 command operation conflict.	
7805		Wait for M code command error.	
7806		Wait for M code inconsistency.	
7807		Servo enable disconnected.	
7810		Absolute encoder homing error.	
7811		Motion axis status error.	
7812		Bus or encoder error during homing.	
7813		Axis status error during torque control.	
7814		Axis value error during torque control.	
7815		Signal error during torque control.	
7816		Detection error during torque control.	
7817		Direction error during torque control.	
7818		Conflict with manual operation during torque control.	
7820		System macro variable write error.	
7821		Local macro variable write error.	
7822		Reference point not established.	
7823		Skip address setting error.	
7830		Rigid tapping signal error.	
7831		Rigid tapping spindle selection error.	

Alarm No.	Significance	Possible Causes	Handling Method
7832		Rigid tapping spindle mode error.	
7833		Drilling spindle selection error.	
7834		Drilling spindle speed not reached.	
7900		Failed to read program line when loading graphic trajectory program.	
7906		A program line exceeds 256 characters when loading graphic trajectory program.	
7907		Address not found when loading graphic trajectory program.	
7908		No data following the address when loading graphic trajectory program.	
7909		Incorrect use of minus sign when loading graphic trajectory program.	
7910		Incorrect use of decimal point when loading graphic trajectory program.	
7911		Illegal address input when loading graphic trajectory program.	
7912		Incorrect G code when loading graphic trajectory program.	
7913		Address duplication error when loading graphic trajectory program.	
7914		Record end character (%) specified or program end statement not specified when loading graphic trajectory program.	
7915		Commanded axis address does not exist in the program when loading graphic trajectory program.	
7920		Bracket nesting error when loading graphic trajectory program.	
7921		Macro statements specified when loading graphic trajectory program.	
7922		Multiple M commands in the same block when loading graphic trajectory program.	

Alarm No.	Significance	Possible Causes	Handling Method
7923		Graphic trajectory program does not end with M99.	
7924		The first coordinate data block of the graphic trajectory program does not specify a rotary axis or specifies multiple rotary axes.	
7925		The axis attribute of the specified rotary axis in the graphic trajectory program is not 0.	
7926		G codes other than G01 are specified in the graphic trajectory program.	
7950		Follow-up program conflicts with main program.	
7951		Follow-up axis servo enable disconnected.	
8001	MST code completion signal output error.	The completion signal of the ladder diagram is detected not to be reset when executing the MST code end.	The ladder diagram needs to be modified.
8002	Waiting M code mismatch.	The main program and the auxiliary axis program specify different waiting M codes.	Please modify the program.
8003	Servo enable disconnected.	Axis movement is commanded when the servo enable is disconnected in the auxiliary program.	Please modify the program.
8010	Illegal variable number used in macro program.	A value that cannot be specified is designated as a variable number in the user macro program.	The program needs to be modified.
8011	Macro variable write protection.	This macro variable has been set to prohibit writing.	Please check parameters No.6031 and No.6032.
8012	Macro variable write prohibition.	This macro variable is read-only and cannot be written to.	The program needs to be modified.
8013	Writing to system macro variables cannot be empty.	When assigning values to system macro variables, empty values cannot be assigned.	The program needs to be modified.
8014	Macro variable data exceeds allowed value range.	The macro variable setting value exceeds the allowed value range.	The data needs to be modified.

Alarm No.	Significance	Possible Causes	Handling Method
8020	Servo working mode error in torque control.	For example, a torque command is received in position mode or a position command is received in torque mode.	Press the [Reset] key to clear the alarm.
8021	Commanded torque value out of range.	The commanded torque value exceeds the value set in servo parameter No.177.	Please modify the program or servo parameters.
8022	F or S value under commanded torque out of range.	The F or S value under commanded torque exceeds the value set in servo parameter 83.	Please modify the program or servo parameters.
8023	Detection error occurred in torque command.	During the operation of the torque command, the torque arrival signal is not detected within the set stroke.	Please modify the program.
8024	Detection error occurred in torque command.	During the operation of the torque command, a torque limit signal is detected.	Please check the machine tool signals.
8025	Direction error occurred in torque command.	During the operation of the torque command, the set detection direction is inconsistent with the actual operation direction.	Please modify the program.
8030	Spindle encoder value error due to unsuccessful PC signal detection.	The drive unit is not connected, or the bus is disconnected, or the spindle is not rotated in place, resulting in failure to correctly obtain encoder readings.	Please check the bus connection or rotate the spindle.
8500	Reference point not established.	It is required to manually move to the reference point, then press the axis movement key in the reference point return mode to establish the reference point.	
8501	Encoder data error.	Encoder data reading error.	The machine reference point needs to be re-established.
8510	Large deviation alarm of power-on machine coordinate system.	The error value between the machine coordinate established on power-on and the memorized machine coordinate exceeds the allowable error. Reasons:1. The carriage position moved during machine power-off;2. The setting value of parameter No.8419 is too small.	The machine reference point needs to be reset or power on again.
8511	Excessively large deviation alarm of power-on machine coordinate system.	The error value between the machine coordinate established on power-on and the memorized machine coordinate is excessively large. Reasons:1. The carriage position moved during machine power-off;2. The motor encoder has been replaced.	Please power on again or reset the machine reference point.

Alarm No.	Significance	Possible Causes	Handling Method
8512	System parameters related to reference point have been changed.	Power-on detection finds that the system parameters are different from those when the machine reference point was established, including parameters No.8407#2, No.8417, or No.8418.	The machine reference point needs to be re-established.
8513	Servo parameters related to reference point have been changed.	Power-on detection finds that the servo parameters are different from those when the machine reference point was established, including command inversion parameters or gear ratio parameters.	The machine reference point needs to be re-established.
8514	Failed to read parameter record file related to reference point.	Power-on detection cannot find the parameter values from the last successful establishment of the machine zero point.	The machine reference point needs to be reset.
8520	Unsuccessful PC signal detection due to unconnected drive unit or disconnected bus.		Please check the bus connection or power on again and try.
8521	Torque control switching error.	The reference point of the absolute encoder of the axis is not established during torque switching.	Please establish the reference point first.
8604	Servo alarm.	Digital servo system failure.	Please check the servo.
8700	Positive travel limit 1.	Exceeded stored travel detection 1 when moving in the positive direction.	Please modify parameter No.8440.
8701	Negative travel limit 1.	Exceeded stored travel detection 1 when moving in the negative direction.	Please modify parameter No.8441.
8702	Positive travel limit 2.	Exceeded stored travel detection 2 when moving in the positive direction.	Please modify parameter No.8442.
8703	Negative travel limit 2.	Exceeded stored travel detection 2 when moving in the negative direction.	Please modify parameter No.8443.