Smarter. Greener. Together.

Industrial Automation Headquarters

Delta Electronics, Inc.
Taoyuan Technology Center
No.18, Xinglong Rd., Taoyuan District,
Taoyuan City 33068, Taiwan
TEL: 886-3-362-6301 / FAX: 886-3-371-6301

Asia

Delta Electronics (Shanghai) Co., Ltd.
No.182 Minyu Rd., Pudong Shanghai, P.R.C.
Post code : 201209
TEL: 86-21-6872-3988 / FAX: 86-21-6872-3996
Customer Service: 400-820-9595

Delta Electronics (Japan), Inc.
Tokyo Office
Industrial Automation Sales Department
2-1-14 Shibadaimon, Minato-ku
Tokyo, Japan 105-0012
TEL: 81-3-5733-1155 / FAX: 81-3-5733-1255

Delta Electronics (Korea), Inc.
Seoul Office
1511, 219, Gasan Digital 1-Ro., Geumcheon-gu,
Seoul, 08501 South Korea
TEL: 82-2-515-5305 / FAX: 82-2-515-5302

Delta Energy Systems (Singapore) Pte Ltd.
4 Kaki Bukit Avenue 1, #05-04, Singapore 417939
TEL: 65-6747-5155 / FAX: 65-6744-9228

Delta Electronics (India) Pvt. Ltd.
Plot No.43, Sector 35, HSIIDC Gurgaon,
PIN 122001, Haryana, India
TEL: 91-124-4874900 / FAX : 91-124-4874945

Delta Electronics (Thailand) PCL.
909 Soi 9, Moo 4, Bangpoo Industrial Estate (E.P.Z),
Pattana 1 Rd., T.Phraksa, A.Muang,
Samutprakarn 10280, Thailand
TEL: 66-2709-2800 / FAX : 662-709-2827

Delta Energy Systems (Australia) Pty Ltd.
Unit 20-21/45 Normanby Rd., Notting Hill Vic 3168, Australia
TEL: 61-3-9543-3720

Americas

Delta Electronics (Americas) Ltd.
Raleigh Office
P.O. Box 12173, 5101 Davis Drive,
Research Triangle Park, NC 27709, U.S.A.
TEL: 1-919-767-3813 / FAX: 1-919-767-3969

Delta Greentech (Brasil) S/A
São Paulo Office
Rua Itapeva, 26 – 3˚ Andar - Bela Vista
CEP: 01332-000 – São Paulo – SP - Brasil
TEL: 55-11-3530-8642 / 55-11-3530-8640

Delta Electronics International Mexico S.A. de C.V.
Mexico Office
Vía Dr. Gustavo Baz No. 2160, Colonia La Loma,
54060 Tlalnepantla Estado de Mexico
TEL: 52-55-2628-3015 #3050/3052

EMEA

Headquarters: Delta Electronics (Netherlands) B.V.
Sales: Sales.IA.EMEA@deltaww.com
Marketing: Marketing.IA.EMEA@deltaww.com
Technical Support: iatechnicalsupport@deltaww.com
Customer Support: Customer-Support@deltaww.com
Service: Service.IA.emea@deltaww.com
TEL: +31(0)40 800 3800

BENELUX: Delta Electronics (Netherlands) B.V.
De Witbogt 20, 5652 AG Eindhoven, The Netherlands
Mail: Sales.IA.Benelux@deltaww.com
TEL: +31(0)40 800 3800

DACH: Delta Electronics (Netherlands) B.V.
Coesterweg 45, D-59494 Soest, Germany
Mail: Sales.IA.DACH@deltaww.com
TEL: +49(0)2921 987 0

France: Delta Electronics (France) S.A.
ZI du bois Challand 2, 15 rue des Pyrénées,
Lisses, 91090 Evry Cedex, France
Mail: Sales.IA.FR@deltaww.com
TEL: +33(0)1 69 77 82 60

Iberia: Delta Electronics Solutions (Spain) S.L.U
Ctra. De Villaverde a Vallecas, 265 1º Dcha Ed.
Hormigueras – P.I. de Vallecas 28031 Madrid
TEL: +34(0)91 223 74 20
Carrer Llacuna 166, 08018 Barcelona, Spain
Mail: Sales.IA.Iberia@deltaww.com

Italy: Delta Electronics (Italy) S.r.l.
Ufficio di Milano Via Senigallia 18/2 20161 Milano (MI)
Piazza Grazioli 18 00186 Roma Italy
Mail: Sales.IA.Italy@deltaww.com
TEL: +39 02 64672538

Russia: Delta Energy System LLC
Vereyskaya Plaza II, office 112 Vereyskaya str.
17 121357 Moscow Russia
Mail: Sales.IA.RU@deltaww.com
TEL: +7 495 644 3240

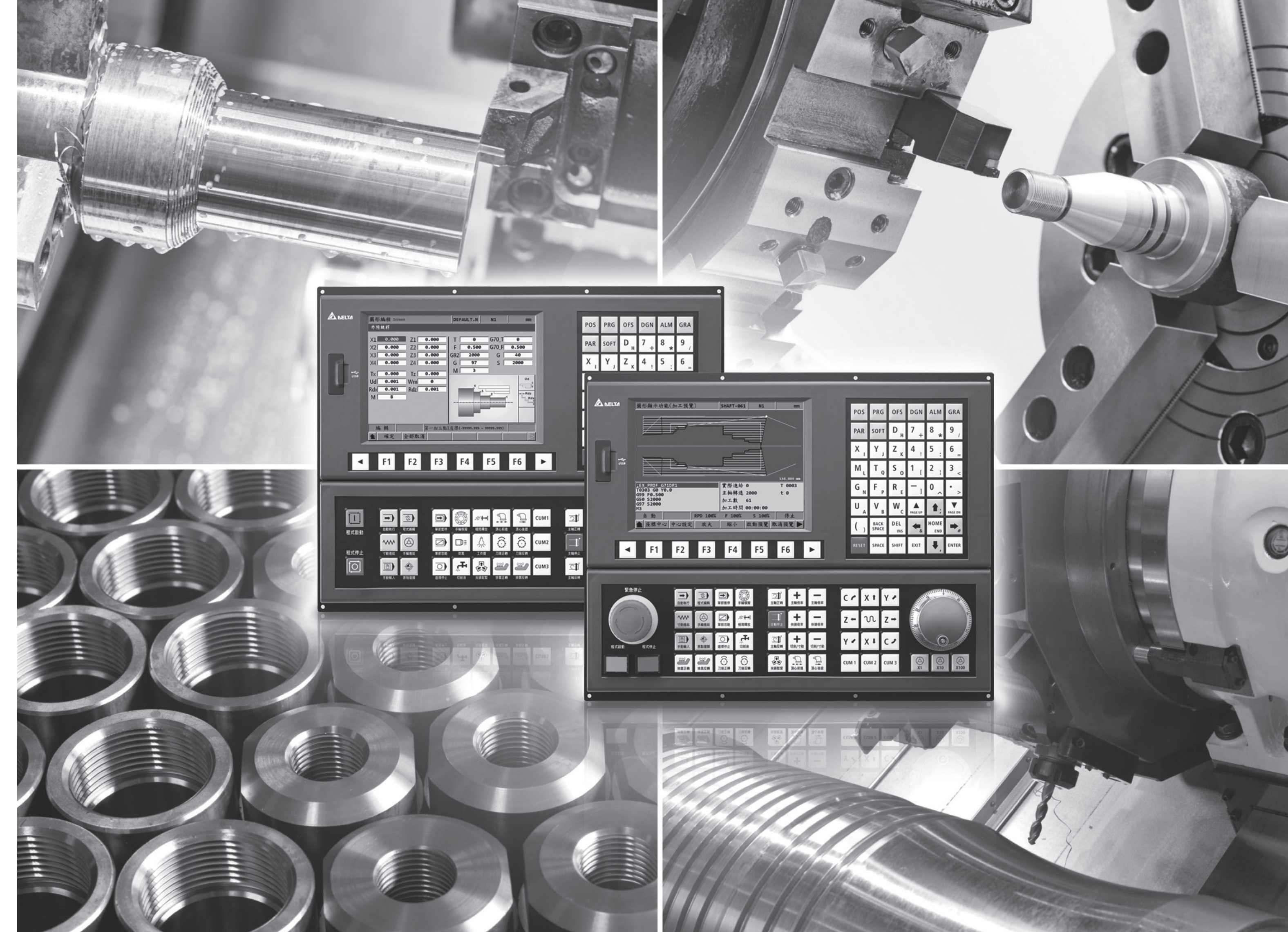
Turkey: Delta Greentech Elektronik San. Ltd. Sti. (Turkey)
Şerifali Mah. Hendem Cad. Kule Sok. No:16-A
34775 Ümraniye – İstanbul
Mail: Sales.IA.Turkey@deltaww.com
TEL: + 90 216 499 9910

GCC: Delta Energy Systems AG (Dubai BR)
P.O. Box 185668, Gate 7, 3rd Floor, Hamarain Centre
Dubai, United Arab Emirates
Mail: Sales.IA.MEA@deltaww.com
TEL: +971(0)4 2690148

Egypt + North Africa: Delta Electronics
511 Cairo Business Plaza, North 90 street,
New Cairo, Cairo, Egypt
Mail: Sales.IA.MEA@deltaww.com

DELTA_IA-CNC_Lathe application_OM_MM_EN_20200430

Delta CNC Lathe Machine Solution Operation and Maintenance Manual



Delta CNC Lathe Machine Solution Operation and Maintenance Manual

www.deltaww.com

Smarter. Greener. Together.

Preface

Thank you for purchasing this product. Read this manual carefully before using the product to ensure the correct use of the product. Keep this manual handy for quick reference whenever needed.

This manual includes:

- Installation and inspection of NC controllers
- Wiring for connectors of the NC controllers
- Description of the function and operation of NC controllers
- Description of parameters
- Troubleshooting

NC controller product features:

- Built-in 32-bit high-speed dual CPU for multi-task execution, improving operating performance
- User-friendly human machine interface
- Interfaces for auto tuning and gain adjustment of the servo, meeting different machine characteristics
- CNCSoft software for configuring user-defined screens
- Front USB interface (port) for easy data access, data backup, and parameter file copying
- In Spindle mode, you can control the spindle system with DMCNET communication or analog voltage according to the requirements
- Serial I/O modules for flexible I/O configuration

How to use this manual:

Use this manual as a reference when installing, setting up, using, and maintaining the NC controller. Read this manual before using and setting this product.

Delta technical services:

Consult your Delta equipment distributors or Delta Customer Service Center if you encounter any problem.

Safety Precautions

- Refer to the pin assignments when wiring. Ensure the product is properly grounded.
- Do not disassemble the controller, change the wiring, or touch the power supply when the power is on to avoid electric shock.

Pay special attention to the following safety precautions at all times during installation, wiring, operation, maintenance, and examination of the controller.

The symbols of "DANGER", "WARNING" and "STOP" indicate:

DANGER	**Danger. May cause severe or fatal injuries to personnel if the instructions are not followed.**
WARNING	**Warning. May cause moderate injury to personnel, or lead to severe damage or even malfunction of the product if the instructions are not followed.**
STOP	**Absolute prohibited activities. May cause serious damage or even malfunction of the product if the instructions are not followed.**

Installation

DANGER

- Follow the installation instructions in the manual, or it may result in damage to the equipment.
- Do not expose the product to an environment containing vapor, corrosive gas, inflammable gas, or other foreign matter to reduce the risk of electric shock or fire.

Wiring

DANGER

- Connect the ground terminal to class-3 ground system. Ground resistance should not exceed 100 Ω. Improper grounding may result in electric shock or fire.

Operation

WARNING

- Correctly plan the I/O configuration with the MLC editor software, or abnormal operation may occur.
- Before operating the machine, properly adjust the parameter settings, otherwise it may cause abnormal operation or malfunction.
- Ensure you can activate the emergency stop at any time, and avoid operating the machine in unprotected condition.

- Do not change the wiring when the power is on, or it may lead to personal injury caused by electric shock.
- Do not use a sharp-pointed object to touch the panel. Doing this may dent the screen and lead to malfunction of the controller.

Maintenance and Inspection

- When the power is on, do not disassemble the controller panel or touch the internal parts of the controller, or it may cause electric shock.
- Do not touch the wiring terminal within 10 minutes after turning off the power since the residual voltage may cause electric shock.
- Turn off the power before replacing the battery, and check the system settings after the replacement.
- Do not block the ventilation holes when operating the controller since poor ventilation may cause malfunction of the controller.

Wiring Method

- Power supply: use a 24 V_{DC} power supply for the controller and comply with the wire specification when wiring to avoid danger.
- Wire selection: use stranded wires and multi-core shielded-pair wires for signal cables.
- Cable length: the maximum cable length of the signal cable for remote I/O and DMCNET communication is 20 m and the maximum cable length of other signal cables is 10 m.
- The local I/O and remote I/O require an additional 24 V_{DC} power for signal input and output.

Wiring of Communication Circuit

- DMCNET wiring: the wiring materials should comply with the standard specification.
- Make sure the controller and servo drive are firmly connected, or loose cables may cause abnormal operation.

Note: the content of this manual may be revised without prior notice. Download the latest version from Delta’s website at https://www.deltaww.com/.

(This page is intentionally left blank.)

Table of Contents

Before Operation

Operation and Settings

4 Table of Group Function

5 Introduction to NC System Modes

6 Position (POS) Group

7 Program (PRG) Group

8 Offset (OFS) Group

9 Diagnosis (DGN) Group

10 Alarm (ALM) Group

11 Graph (GRA) Group

Parameter Settings

12 Parameter (PAR) Group

Customized Function

13 Software (SOFT) Group

How to check MLC status

14 MLC Special M Relay and Special D Register

Troubleshooting

15 Troubleshooting

Product Inspection and Model Explanation

1

Before using the NC series products, read this chapter for information about the model explanation and product interface.

1

1.1 Product inspection

In case of packaging mistakes or damages during shipping, check the items listed in the following table carefully. If any issue occurs, contact the distributor for service.

Item	Content
Purchased product	Check the model number on the nameplate of the controller. Refer to Section 1.2 Model explanation.
Buttons	Press the buttons to check if the operation is smooth*.
Appearance	Visually check if there are any damages on the exterior of the product.
Connectors	Check if there are any loose or untightened connectors.

Note: check the buttons for all models except for the NC30E(H) series, which does not have operation panels.

1.2 Model explanation

■ Nameplate information

■ Model explanation

NC series controller (MOP integrated)

NC2	0	0	A	–	L	I	–	A	E
(1)	(2)	(3)	(4)		(5)	(6)		(7)	(8)

(1) Series name

NC2: 2 series controller

(2) Display

0: 8” screen

(3) Screen orientation

0: horizontal

(4) Series type

A: A series

P: MPG series (MPG included)

(5) Model

M: milling

L: lathe

(6) Type

I: integrated (machine operation panels integrated)

(7) Version

A: standard

(8) Language

Blank: Traditional Chinese

S: Simplified Chinese

E: English

OPENCNC series controller

NC30 E
(1) (2)

(1) Series name

NC30: 3 series controller

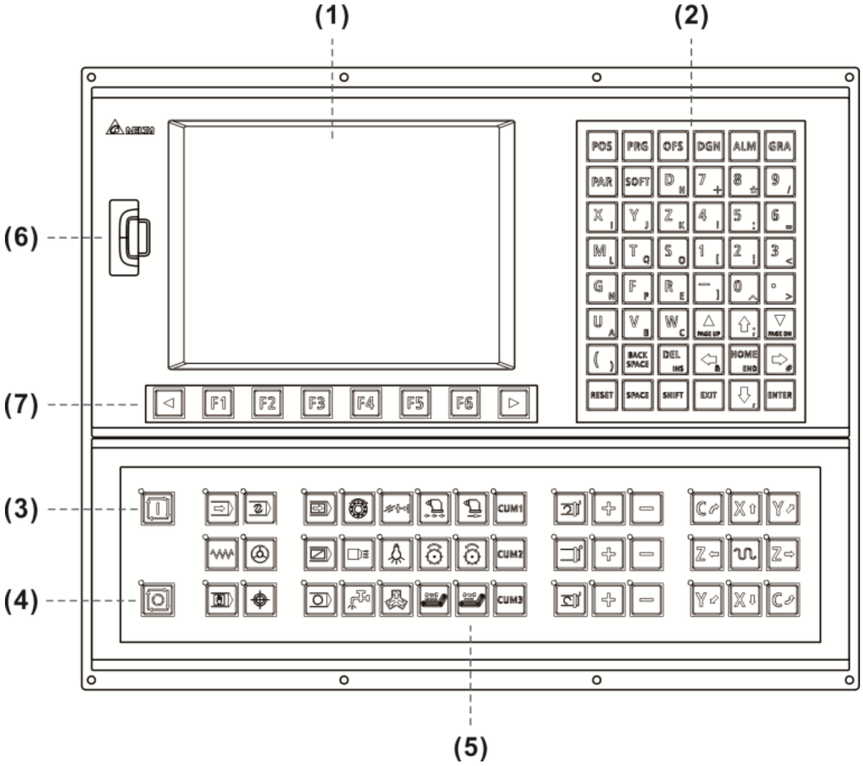
(2) Series type

E: embedded series

EH: embedded multi-axis series

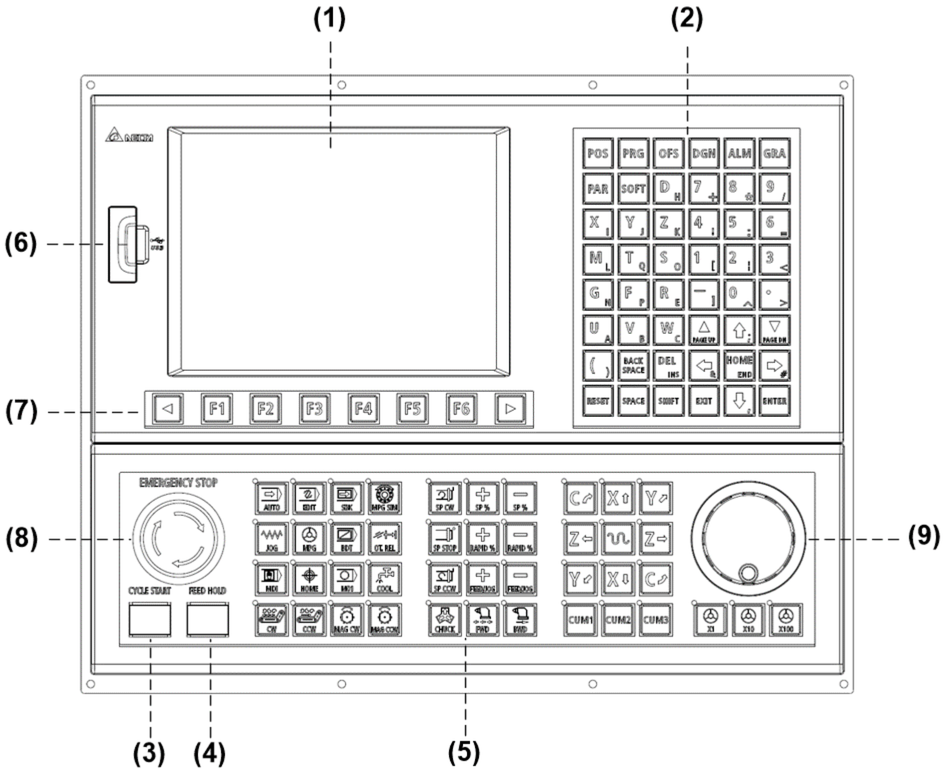
1.3 Product interface of NC controller

NC200A-LI-A_



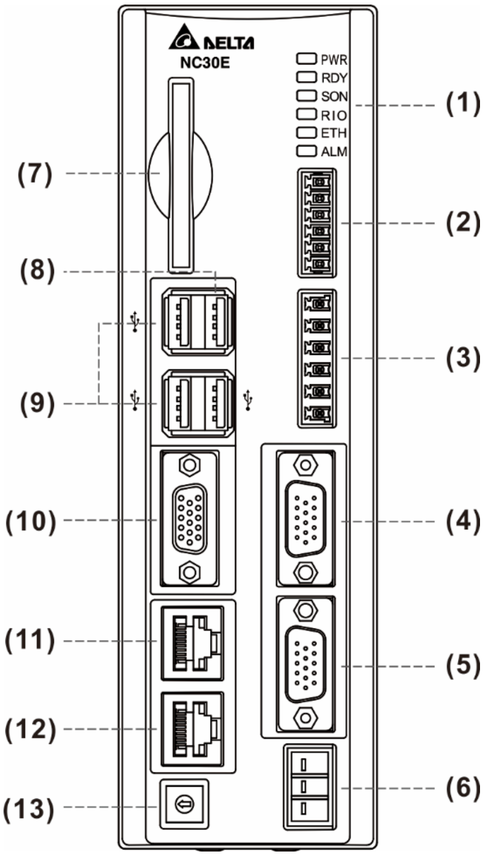
No.	Item	No.	Item
(1)	Screen	(5)	Machine operation panel B
(2)	Machine operation panel A	(6)	USB port
(3)	CYCLE START	(7)	Function keys
(4)	FEED HOLD	-	-

NC200P-LI-A_



No.	Item	No.	Item
(1)	Screen	(6)	USB port
(2)	Machine operation panel A	(7)	Function keys
(3)	CYCLE START	(8)	Emergency stop
(4)	FEED HOLD	(9)	MPG
(5)	Machine operation panel B	-	-

NC30E(H)



No.	Item	No.	Item
(1)	Indicators	(8)	PAN interface
(2)	Remote I/O module connector	(9)	USB ports
(3)	HSI connector	(10)	VGA connector
(4)	MPG connector	(11)	Ethernet connector
(5)	Spindle connector	(12)	DMCNET connector
(6)	24 V_{DC} power input	(13)	Mode switch
(7)	CF card slot	-	-

(This page is intentionally left blank.)

Installation

2

Follow the precautions and storage and installation conditions specified in this chapter when installing the product. In addition, outline dimensions and hardware specifications of the product are provided.

2.1 Precautions

Follow the precautions and specifications of voltage, current, temperature, and other conditions specified in the manual, or personnel injury and equipment damage may occur.

2.2 Ambient storage conditions

Before installation, this product must be kept in the shipping carton. To retain the warranty coverage and ensure future maintenance, follow these instructions for storage. While the product is temporarily not in use:

- Store the product in a dust-free and dry location.
- Store the product in an ambient temperature range of -20°C to +60°C (-4°F to +140°F).
- Store the product in a relative humidity range of 10% to 95% RH (non-condensing).
- Avoid storing the product in an environment containing corrosive gas and liquids.
- The environment should be free of devices that generate excessive heat; no water, vapor, dust, and oily dust; no corrosive and inflammable gas or liquids; no airborne dust or metal particles; the environment should be solid without vibration and interference of electromagnetic noise.
- Hardware specifications:

NC2 series lathe controller

Model	NC200A-LI-A_	NC200P-LI-A_
Operating environment	10% to 95% RH [0°C to +55°C (32°F to 131°F)]	
Storage environment	10% to 95% RH [-20°C to +60°C (-4°F to +140°F)]	
Cooling method	Natural cooling	
Voltage	24 V_{DC} (-10% to +15%) (built-in isolated circuit)	
Dielectric withstanding voltage	Between 24 V_{DC} and FG terminals: 500 V_{AC} for 1 minute	
Power consumption	15 W (24V; 0.6 A)	
Memory backup battery	3V lithium battery (CR2032) × 1	
Backup battery life	Varies according to the ambient temperature and operating conditions; approximately 3 years in room temperature of 25°C (77°F)	
Dimensions	Refer to the outline dimension table in Section 2.5.	
Weight	4.5 kg	4.7 kg

OPENCNC series controller

Model	NC30E(H)
Operating environment	10% to 95% RH [0°C to +55°C (32°F to 131°F)]
Storage environment	10% to 95% RH [-20°C to +60°C (-4°F to +140°F)]
Cooling method	Natural cooling
Voltage	24 V_{DC} (-10% to +15%) (built-in isolated circuit)
Dielectric withstanding voltage	Between 24 V_{DC} and FG terminals: 500 V_{AC} for 1 minute
Power consumption	15 W (24V; 0.6 A)
Memory backup battery	3V lithium battery (CR2032) × 1
Backup battery life	Varies according to the ambient temperature and operating conditions; approximately 3 years in room temperature of 25°C (77°F)
Dimensions	Refer to the outline dimension table in Section 2.5.
Weight	0.8 kg

2.3 Ambient installation conditions

2

Operating temperature for the NC series controller: 0°C - 55°C (32°F - 131°F).

During long-term operation, the suggested temperature of the operating environment should be below 45°C (113°F). If the temperature is above 45°C, place the product in a distribution board which is well-ventilated and without overheating risks. Also check if the vibration of the machine affects the electrical devices in the distribution board.

In addition, follow these precautions when choosing the installation site to retain the warranty coverage and ensure future maintenance for the Delta NC controller.

- The environment should be free of devices that generate excessive heat; no water, vapor, dust, and oily dust; no corrosive and inflammable gas or liquids; no airborne dust or metal particles; the environment should be solid without vibration and interference of electromagnetic noise.
- The temperature and humidity of the installation site for the NC controller should be within the range specified in the specification.
- Avoid storing the NC controller in an environment where the vibration exceeds the range specified in the specification.

2.4 Mounting direction and space

The NC series controller must be installed vertically on a dry and solid platform which complies with the requirement of NEMA. For better ventilation and cooling, allow sufficient clearance space between the controller and its adjacent objects and the wall (the clearance is suggested to be 50 mm (around 2 inches)).

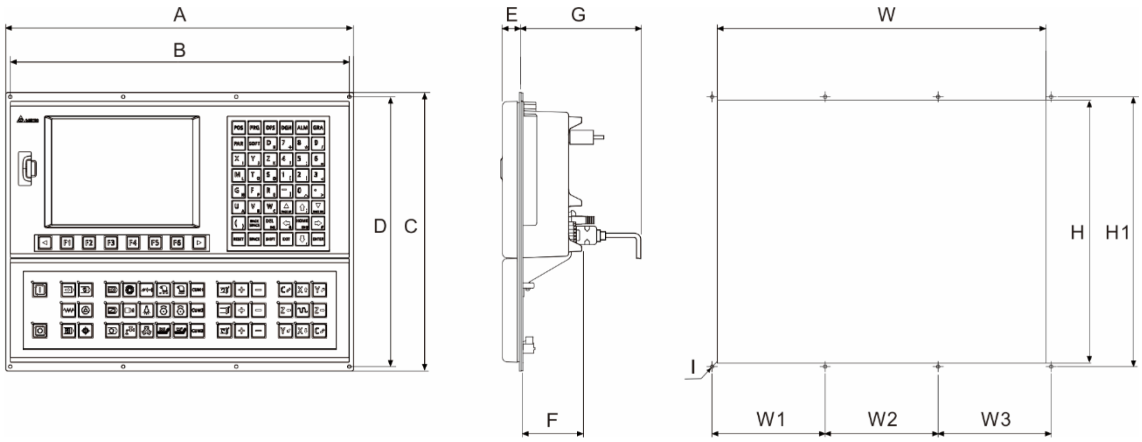
Important:

- For better ventilation and cooling, allow sufficient clearance space between the NC controller and the adjacent objects and the wall, or it may cause malfunction of the machine.
- Do not block the ventilation holes of the NC controller, or it may cause malfunction of the machine.

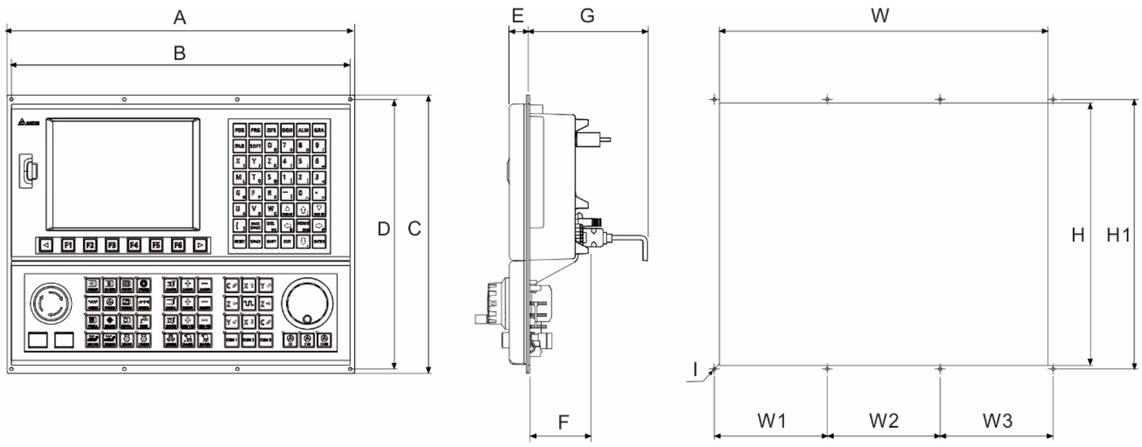
2.5 Outline and mounting dimensions

Appearance and dimension tables of NC2 lathe series

NC200-A-LI-A_ dimension drawing



NC200-P-LI-A_ dimension drawing



Outline dimension table

Model / Dimension	NC200A-LI-A_	NC200P-LI-A_
A	400 mm	400 mm
B	390 mm	390 mm
C	320 mm	320 mm
D	310 mm	310 mm
E	22 mm	22 mm
F	70 mm	70 mm
G	130 mm*	130 mm*

Note: this is the minimum required space for installation.

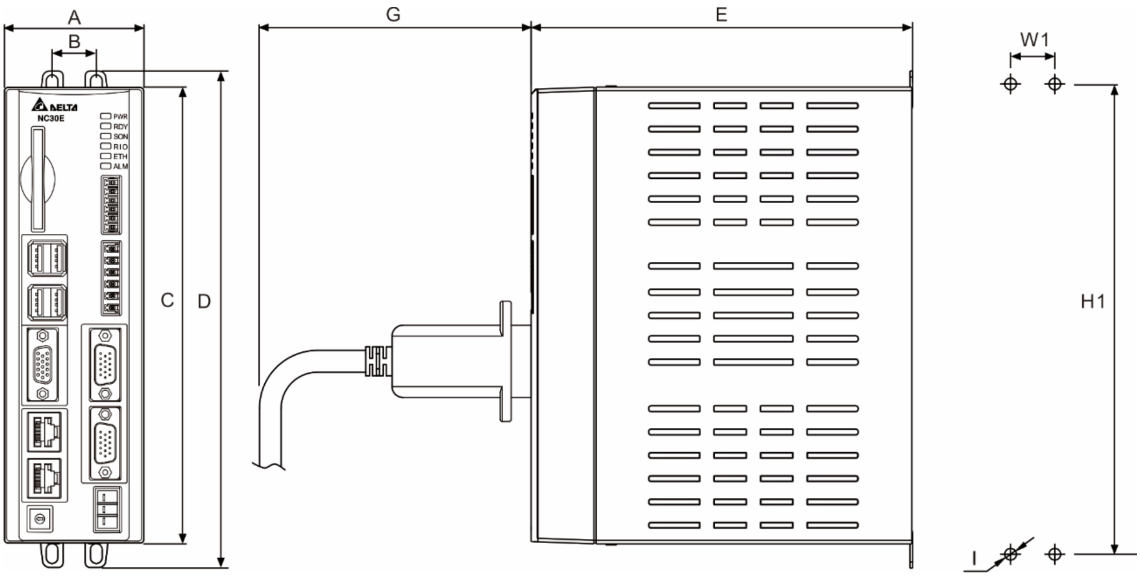
Mounting dimension table

Model / Dimension	NC200A-LI-A_	NC200P-LI-A_
H	302 ± 0.3 mm	302 ± 0.3 mm
H1	310 mm	310 mm
W	378 ± 0.3 mm	378 ± 0.3 mm
W1	130 mm	130 mm
W2	130 mm	130 mm
W3	130 mm	130 mm
I	Φ4 mm	Φ4 mm

2

Appearance and dimension tables of OPENCNC series

NC30E(H) dimension drawing



Outline dimension table

Dimension \ Model	NC30E(H)
A	60 mm
B	19 mm
C	196 mm
D	213 mm
E	164 mm
G	70 mm

Mounting dimension table

Dimension \ Model	NC30E(H)
H1	201 ± 0.3 mm
W1	19 ± 0.3 mm
I	M5

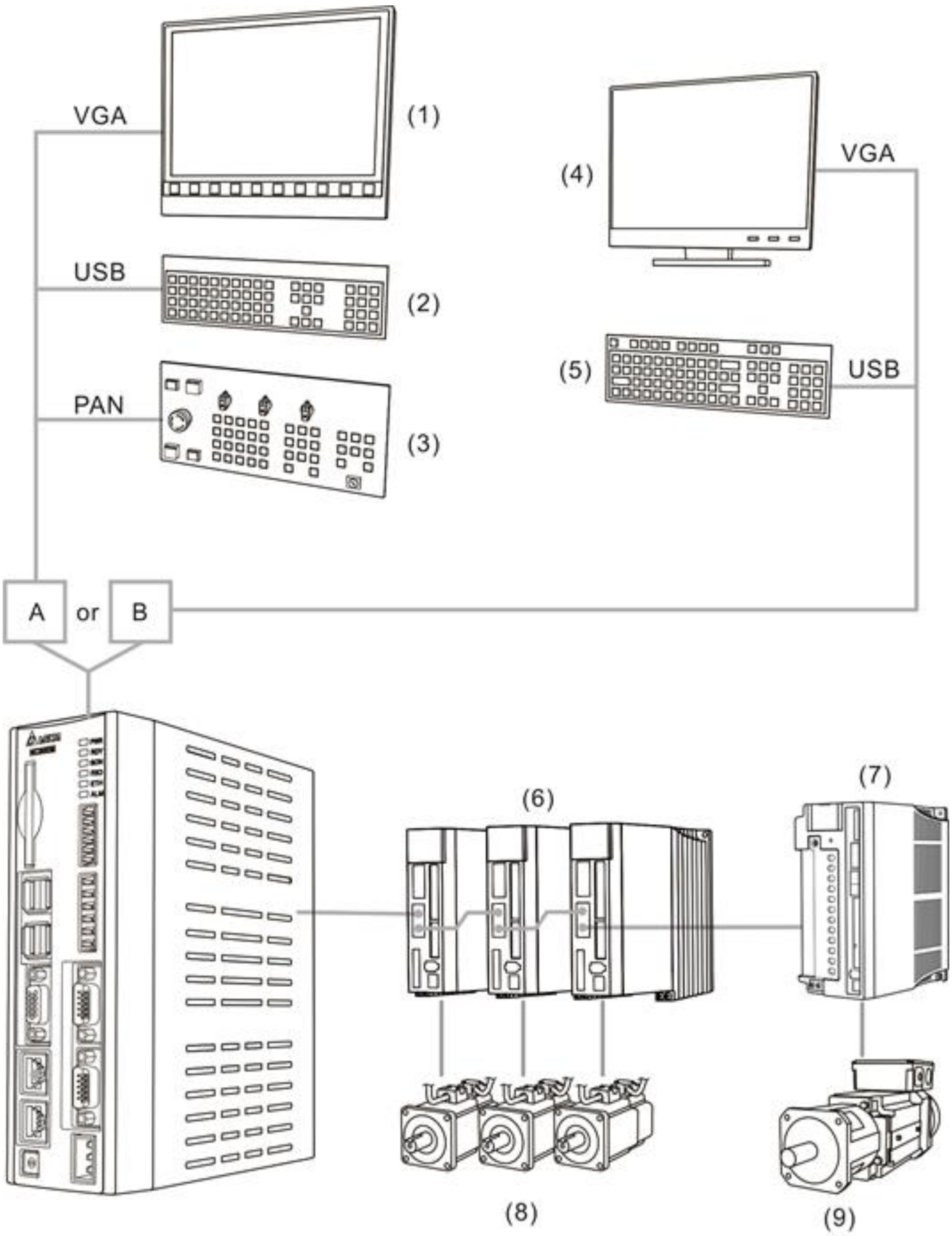
2

2.6 OPENCNC operating interface installation

- Operating interface installation

A. Connect the OPENCNC series controller to standard operating interfaces, including (1) Delta HMI screen, (2) keyboard (functions the same as machine operation panel A), and (3) machine operation panel B.

B. Connect the OPENCNC series controller to non-Delta operating interfaces, including (4) screen and (5) keyboard.

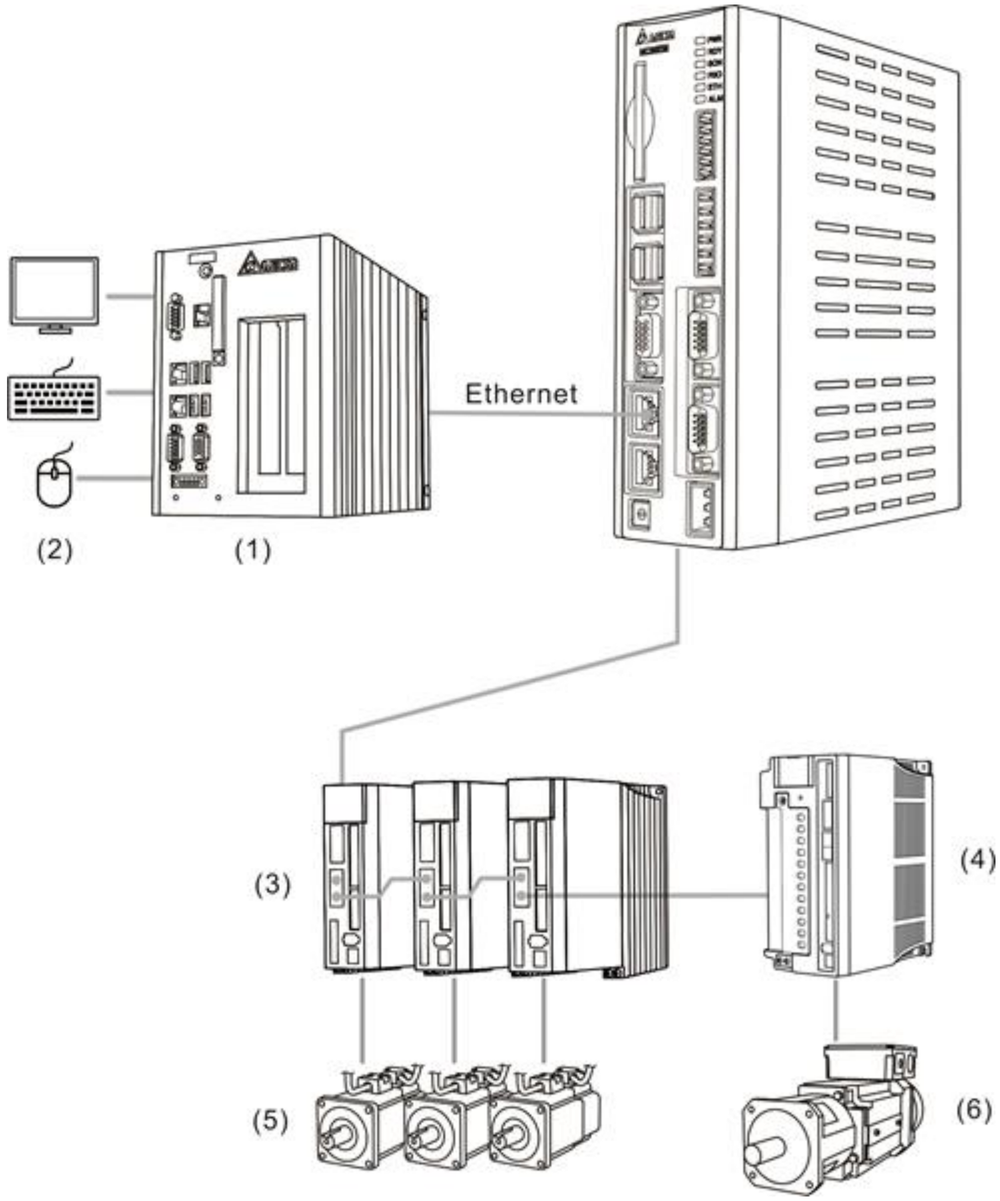


Connect the OPENCNC series controller to (6) ASDA series AC servo drives, (7) ASDA-S spindle servo drive, (8) ECMA series servo motors, and (9) ECMS series spindle servo motor.

2

- Connection through computer and network

Connect the OPENCNC series controller to (3) ASDA series AC servo drives, (4) ASDA-S spindle servo drive, (5) ECMA series servo motors, and (6) ECMS series spindle servo motor through the computer and network, including (1) PC-based controller and (2) mouse and the public software for Delta OPENCNC.



(This page is intentionally left blank.)

Wiring

3

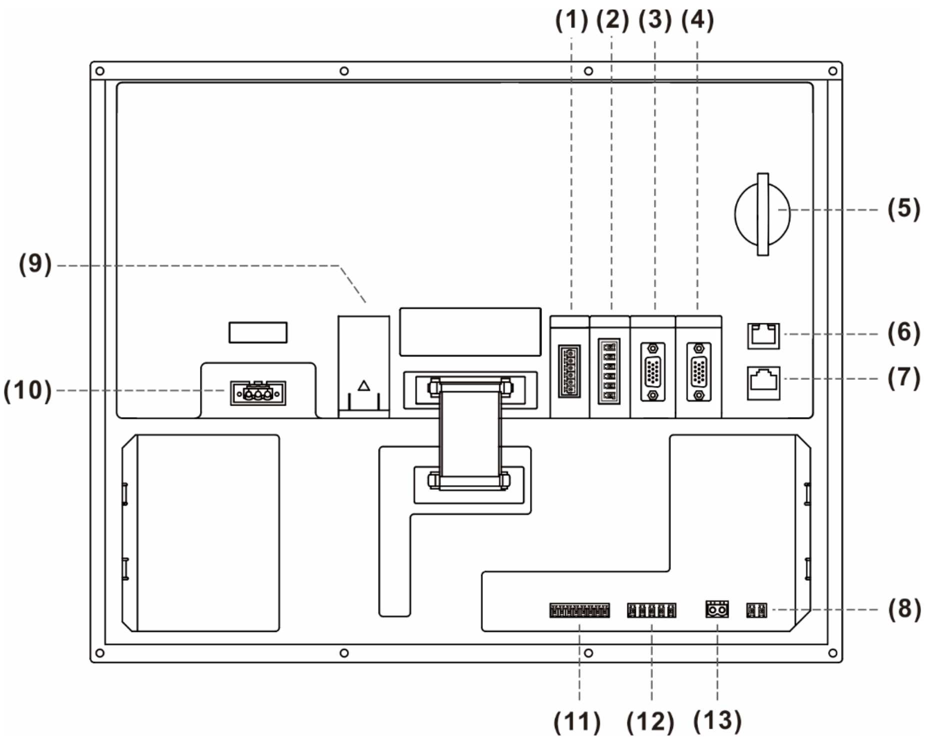
This chapter illustrates the wiring and connectors of the controller, and provides the wiring diagrams for each function.

3.1 Product interface

■ NC2 series

3

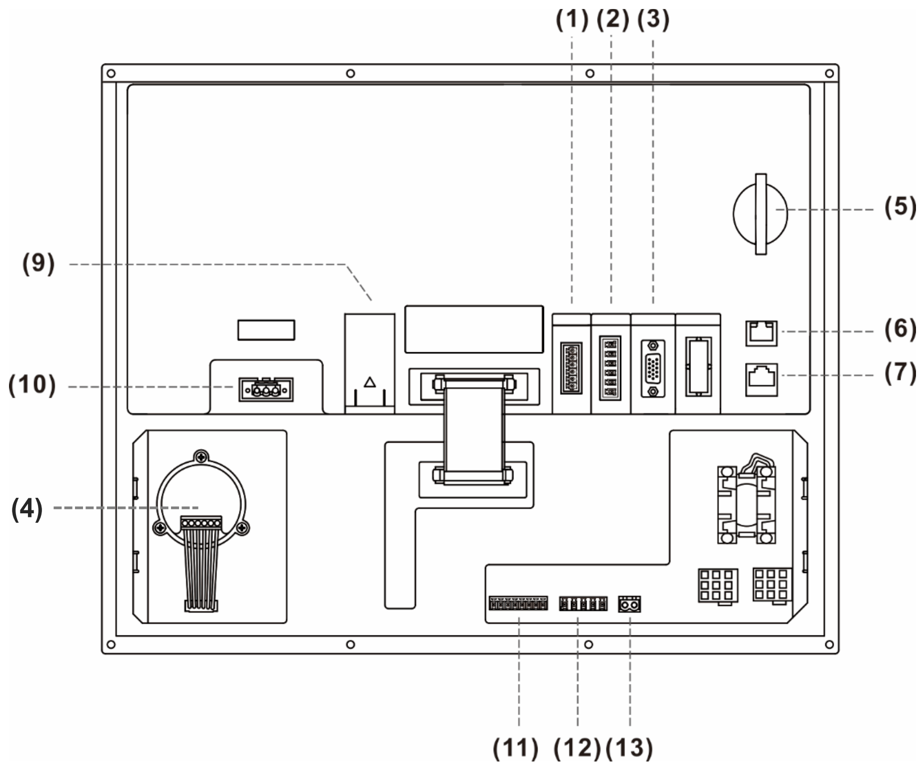
NC200A-LI-A_



Description of each connector:

No.	Connector	Description	
(1)	Remote I/O	Connects to high-speed serial I/O module: each module has 32 DI and 32 DO points. The maximum distance between stations is 20 m (65.62 ft) and the total length can be up to 160 m (20 m x 8) (524.94 ft).	
(2)	HSI	High-speed counter and emergency stop input signal.	
(3)	Spindle	One set of spindle encoder input signal.	
		One set of analog output signal.	
		One set of RS-485 serial communication signal.	
(4)	MPG	Connects to an external MPG (powered by the built-in 5 V_{DC} power). 7 DI and 1 DO points are available.	
(5)	CF card	For storing G-code programs.	
(6)	Ethernet	For DNC control and system monitoring.	
(7)	DMCNET	High-speed communication network interface.	
(8)	Emergency stop	Only available on NC200A-LI-A models.	
(9)	-	Battery holder.	
(10)	24 V_{DC} power	24 V_{DC} power input for supplying power to the controller.	
(11)	Local I/O	LI series	DI: 8 input points.
(12)			DO: 5 output points.
(13)	24 V_{DC} power	For supplying power to the local I/O.	

NC200P-LI-A_



Description of each connector:

No.	Connector	Description	
(1)	Remote I/O	Connects to high-speed serial I/O module: each module has 32 DI and 32 DO points. The maximum distance between stations is 20 m (65.62 ft) and the total length can be up to 160 m (20 x 8) (524.94 ft).	
(2)	HSI	High-speed counter and emergency stop input signal.	
(3)	Spindle	One set of spindle encoder input signal.	
		One set of analog output signal.	
		One set of RS-485 serial communication signal.	
(4)	-	Embedded MPG of machine operation panel B.	
(5)	CF card	For storing G-code programs.	
(6)	Ethernet	For DNC control and system monitoring.	
(7)	DMCNET	High-speed communication network interface.	
(8)	Emergency stop	Only available on NC200A-LI-A models.	
(9)	-	Battery holder.	
(10)	24 V_{DC} power	24 V_{DC} power input for supplying power to the controller.	
(11)	Local I/O	LI series	DI: 8 input points.
(12)			DO: 5 output points.
(13)	24 V_{DC} power	For supplying power to the local I/O and the lights for CYCLE START and FEED HOLD buttons.	

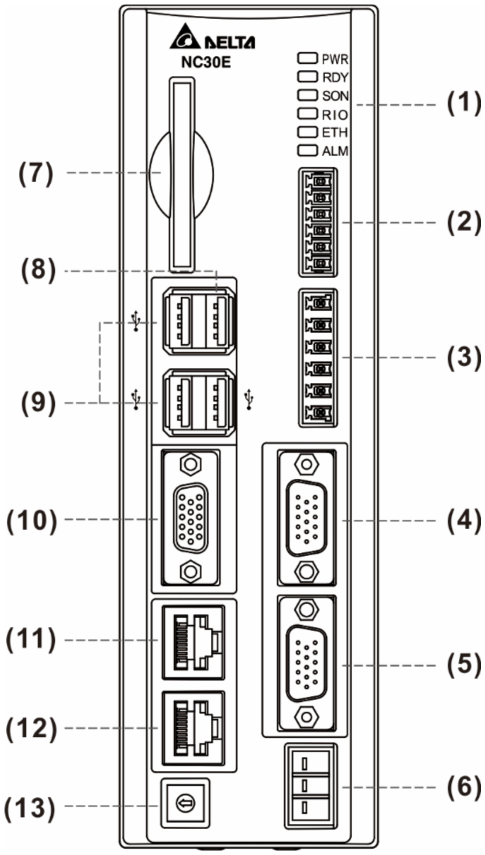
Safety precautions for installation:

1. Check if the wiring for 24 V_{DC} power is correct.
2. Remote I/O requires an additional 24 V_{DC} power supply to drive X input and Y output.
3. Short-circuit IES (emergency stop) to have the controller ready for use.
4. If an alarm occurs or the emergency stop signal is on, power off the servo drive by disconnecting the power at the magnetic contactor (MC) with Y output.

3

■ OPENCNC series

NC30E(H)



Description of each connector:

No.	Connector	Description
(1)	-	Indicators.
(2)	Remote I/O	Connects to high-speed serial I/O module: each module has 32 DI and 32 DO points. The maximum distance between stations is 20 m (65.62 ft) and the total length can be up to 160 m (20 m x 8) (524.94 ft).
(3)	HSI	High-speed counter and emergency stop input signal.
(4)	MPG	Connects to an external MPG (powered by the built-in 5 V_{DC} power). 8 DI points are available.
(5)	Spindle	One set of spindle encoder input signal.
		One set of analog output signal.
		One set of RS-485 serial communication signal.
(6)	24 V_{DC} power	24 V_{DC} power input for supplying power to the controller.
(7)	CF card	For storing G-code programs.
(8)	PAN	One USB connector for connecting to machine operation panel B.
(9)	USB	Three USB host connectors for connecting to the mouse, keyboard, and flash drive.
(10)	VGA	For connecting to a screen.
(11)	Ethernet	Ethernet interface.
(12)	DMCNET	High-speed communication network interface.
(13)	-	Mode switch for switching the debugging mode.

Debugging mode:

<table>
<tr><th>Setting value</th><th>Mode</th></tr>
<tr><td>0</td><td>Standard mode
Indicator description:
<table>
<tr><td>PWR</td><td>Green: system power on</td></tr>
<tr><td>RDY</td><td>Flashing green: system ready
Orange: in modes 1 - 3</td></tr>
<tr><td>SON</td><td>Green: system ready</td></tr>
<tr><td>RIO</td><td>Green: remote I/O connection is normal</td></tr>
<tr><td>ETH</td><td>Green: network communication is normal</td></tr>
<tr><td>ALM</td><td>Red: system alarm occurs
Flashing green: software updating</td></tr>
</table></td></tr>
<tr><td>1</td><td>System update mode</td></tr>
<tr><td>2</td><td>Default mode</td></tr>
<tr><td>3</td><td>Reset IP to default setting
Default IP:
<table>
<tr><td>Controller IP</td><td>192.168.1.11</td></tr>
<tr><td>Subnet mask</td><td>255.255.255.0</td></tr>
<tr><td>Remote PC IP</td><td>192.168.1.10</td></tr>
</table></td></tr>
</table>

Safety precautions for installation:

1. Check if the wiring for 24 V_{DC} power is correct.
2. Remote I/O requires an additional 24 V_{DC} power supply to drive X input and Y output.
3. If an alarm occurs or the emergency stop signal is on, power off the servo drive by disconnecting the power at the magnetic contactor (MC) with Y output.

3.2 Connectors of the controller

3.2.1 NC2 series connectors

<table>
<tr><th>Symbol</th><th>Function</th><th>Description</th></tr>
<tr><td>0V, +24V, ⏚</td><td>Power input for controller</td><td>Connects to 24 V_{DC} power (15 W at 0.6 A).
<table>
<tr><th>Symbol</th><th>Description</th></tr>
<tr><td>+24V</td><td>+24 V_{DC} power</td></tr>
<tr><td>0V</td><td>0 V_{DC} power</td></tr>
<tr><td>⏚</td><td>Power grounding</td></tr>
</table></td></tr>
<tr><td>0V, +24V</td><td>Power input for machine operation panel B</td><td>Connects to 24 V_{DC} power (15 W at 0.6 A).
<table>
<tr><th>Symbol</th><th>Description</th></tr>
<tr><td>+24V</td><td>+24 V_{DC} power</td></tr>
<tr><td>0V</td><td>0 V_{DC} power</td></tr>
</table></td></tr>
<tr><td rowspan="2">I/O</td><td>DI</td><td>Power specifications for DI points:
voltage < 24 V_{DC}, current: 8 - 25 mA.
<table>
<tr><th>Pin No.</th><th>Description</th></tr>
<tr><td>X112 - X119</td><td>DI points X112 - X119, 8 points in total.</td></tr>
</table></td></tr>
<tr><td>DO</td><td>Power specifications for DO points:
voltage < 24 V_{DC}, current < 120 mA.
<table>
<tr><th>Pin No.</th><th>Description</th></tr>
<tr><td>Y112 - Y116</td><td>DO points Y112 - Y116, 5 points in total.</td></tr>
</table></td></tr>
<tr><td>SPINDLE</td><td>Spindle connector</td><td>Includes signals of spindle feedback, analog output, and 1 set of RS-485 serial communication.
<table>
<tr><th>Pin No.</th><th>Description</th></tr>
<tr><td>P1</td><td>RS485_GND</td></tr>
<tr><td>P2</td><td>RS485_D+</td></tr>
<tr><td>P3</td><td>RS485_D-</td></tr>
<tr><td>P4</td><td>SP_OUT</td></tr>
<tr><td>P5</td><td>SP_GND</td></tr>
<tr><td>P6</td><td>Reserved</td></tr>
<tr><td>P7</td><td>EMG_IN; connects to +24 V_{DC}</td></tr>
<tr><td>P8</td><td>SP_A+</td></tr>
<tr><td>P9</td><td>SP_A-</td></tr>
<tr><td>P10</td><td>DC +5V_OUT</td></tr>
<tr><td>P11</td><td>SP_B+</td></tr>
<tr><td>P12</td><td>SP_B-</td></tr>
<tr><td>P13</td><td>SP_Z+</td></tr>
<tr><td>P14</td><td>SP_Z-</td></tr>
<tr><td>P15</td><td>GND</td></tr>
</table></td></tr>
<tr><td>MPG</td><td>MPG connector</td><td>7 DI points, 1 DO point, and 1 set of differential type MPG input signal.
<table>
<tr><th>Pin No.</th><th>Description</th></tr>
<tr><td>P1</td><td>DI_COM; connects to +24 V_{DC} or 0V</td></tr>
<tr><td>P2 - P7</td><td>DI (X28 - X33)</td></tr>
<tr><td>P8</td><td>DI (X26)</td></tr>
<tr><td>P9</td><td>DO (Y27)</td></tr>
<tr><td>P10</td><td>DC +5V_OUT (< 200 mA)</td></tr>
<tr><td>P11</td><td>XA+</td></tr>
<tr><td>P12</td><td>XA-</td></tr>
<tr><td>P13</td><td>XB+</td></tr>
<tr><td>P14</td><td>XB-</td></tr>
<tr><td>P15</td><td>GND</td></tr>
</table></td></tr>
</table>

<table>
<tr><th>Symbol</th><th>Function</th><th colspan="2">Description</th></tr>
<tr><td rowspan="8">HSI</td><td rowspan="8">High-speed counter and emergency stop input signal</td><td colspan="2">2 sets of high-speed and 1 set of emergency stop input signal.</td></tr>
<tr><td>Pin No.</td><td>Description</td></tr>
<tr><td>P1</td><td>EMG_IN</td></tr>
<tr><td>P2</td><td>EMG_GND</td></tr>
<tr><td>P3</td><td>HSI_1 (Counter C78, enter [M2142])</td></tr>
<tr><td>P4</td><td>HSI_COM; connects to +24 V_{DC} or 0V</td></tr>
<tr><td>P5</td><td>HSI_2 (Counter C79, enter [M2143])</td></tr>
<tr><td>P6</td><td>HSI_COM; short-circuit P4 and P6.</td></tr>
<tr><td rowspan="8">REMOTE I/O</td><td rowspan="8">Remote I/O module connector</td><td colspan="2">Remote I/O module communication signals, including X256 - X511 and Y256 - Y511. Each module has 32 DI and 32 DO points, and the controller can connect to up to 8 modules.</td></tr>
<tr><td>Pin No.</td><td>Description</td></tr>
<tr><td>P1</td><td>TX+</td></tr>
<tr><td>P2</td><td>TX-</td></tr>
<tr><td>P3</td><td>RX-</td></tr>
<tr><td>P4</td><td>RX+</td></tr>
<tr><td>P5</td><td>GND</td></tr>
<tr><td>P6</td><td>SHIELD</td></tr>
<tr><td rowspan="10">ETHERNET</td><td rowspan="10">Ethernet connector</td><td colspan="2">Connects to PC with an RJ45 connector and a network cable.
Pin definition of both ends:</td></tr>
<tr><td>Pin No. / color of end A</td><td>Pin No. / color of end B</td></tr>
<tr><td>1: orange & white</td><td>1: orange & white</td></tr>
<tr><td>2: orange</td><td>2: orange</td></tr>
<tr><td>3: green & white</td><td>3: green & white</td></tr>
<tr><td>4: blue</td><td>4: blue</td></tr>
<tr><td>5: blue & white</td><td>5: blue & white</td></tr>
<tr><td>6: green</td><td>6: green</td></tr>
<tr><td>7: brown & white</td><td>7: brown & white</td></tr>
<tr><td>8: brown</td><td>8: brown</td></tr>
<tr><td>DMCNET</td><td>DMCNET connector</td><td colspan="2">Connects to Delta's DMCNET servo drive with standard RJ45 connector. The wiring method is the same as that of the Ethernet connector.</td></tr>
<tr><td>EMG</td><td>Emergency stop</td><td colspan="2">A normally-closed contact. Press to open the circuit in an emergency.</td></tr>
<tr><td>CYCLE START</td><td>CYCLE START button</td><td colspan="2">CYCLE START contact</td></tr>
<tr><td>FEED HOLD</td><td>FEED HOLD button</td><td colspan="2">FEED HOLD contact</td></tr>
<tr><td>IES</td><td>Emergency stop contact</td><td colspan="2">The emergency stop contact. The EMG flag is enabled in a broken circuit. (Wire it to the normally-closed contact of the emergency stop button.)</td></tr>
</table>

Note:

1. The IES connector is the input contact of the emergency stop signal. Enable the EMG flag in a broken circuit.
2. The power for the lights of CYCLE START and FEED HOLD buttons is 24 V_{DC}.

3

3.2.2 OPENCNC series connectors

<table>
<tr><th>Symbol</th><th>Function</th><th colspan="2">Description</th></tr>
<tr><td rowspan="5">0V, +24V, ⏚</td><td rowspan="5">Power input for controller</td><td colspan="2">Connects to the 24 V_{DC} power (15 W at 0.6 A).</td></tr>
<tr><td>Symbol</td><td>Description</td></tr>
<tr><td>+24V</td><td>+24 V_{DC} power</td></tr>
<tr><td>0V</td><td>0 V_{DC} power</td></tr>
<tr><td>⏚</td><td>Power grounding</td></tr>
<tr><td rowspan="17">SPINDLE</td><td rowspan="17">Spindle connector</td><td colspan="2">Includes signals of spindle feedback, analog output, and 1 set of RS-485 serial communication.</td></tr>
<tr><td>Pin No.</td><td>Description</td></tr>
<tr><td>P1</td><td>RS485_GND</td></tr>
<tr><td>P2</td><td>RS485_D+</td></tr>
<tr><td>P3</td><td>RS485_D-</td></tr>
<tr><td>P4</td><td>SP_OUT</td></tr>
<tr><td>P5</td><td>SP_GND</td></tr>
<tr><td>P6</td><td>Reserved</td></tr>
<tr><td>P7</td><td>Reserved</td></tr>
<tr><td>P8</td><td>SP_A+</td></tr>
<tr><td>P9</td><td>SP_A-</td></tr>
<tr><td>P10</td><td>DC +5V_OUT</td></tr>
<tr><td>P11</td><td>SP_B+</td></tr>
<tr><td>P12</td><td>SP_B-</td></tr>
<tr><td>P13</td><td>SP_Z+</td></tr>
<tr><td>P14</td><td>SP_Z-</td></tr>
<tr><td>P15</td><td>GND</td></tr>
<tr><td rowspan="10">MPG</td><td rowspan="10">MPG connector</td><td colspan="2">8 DI points and 1 set of differential type MPG signal input.</td></tr>
<tr><td>Pin No.</td><td>Description</td></tr>
<tr><td>P1</td><td>DI_COM; connects to +24 V_{DC} or 0V</td></tr>
<tr><td>P2 - P9</td><td>DI (X28 - X35)</td></tr>
<tr><td>P10</td><td>DC +5V_OUT (< 200 mA)</td></tr>
<tr><td>P11</td><td>XA+</td></tr>
<tr><td>P12</td><td>XA-</td></tr>
<tr><td>P13</td><td>XB+</td></tr>
<tr><td>P14</td><td>XB-</td></tr>
<tr><td>P15</td><td>GND</td></tr>
<tr><td rowspan="8">HSI</td><td rowspan="8">High-speed counter and emergency stop input signal</td><td colspan="2">2 sets of high-speed and 1 set of emergency stop input signal.</td></tr>
<tr><td>Pin No.</td><td>Description</td></tr>
<tr><td>P1</td><td>EMG_IN (+5 V_{DC} output)</td></tr>
<tr><td>P2</td><td>MODE_ENABLE (+5 V_{DC} output)</td></tr>
<tr><td>P3</td><td>HSI_1 (Counter C78, enter [M2142])</td></tr>
<tr><td>P4</td><td>GND</td></tr>
<tr><td>P5</td><td>HSI_2 (Counter C79, enter [M2143])</td></tr>
<tr><td>P6</td><td>HSI_COM; connects to +24 V_{DC} or 0V</td></tr>
</table>

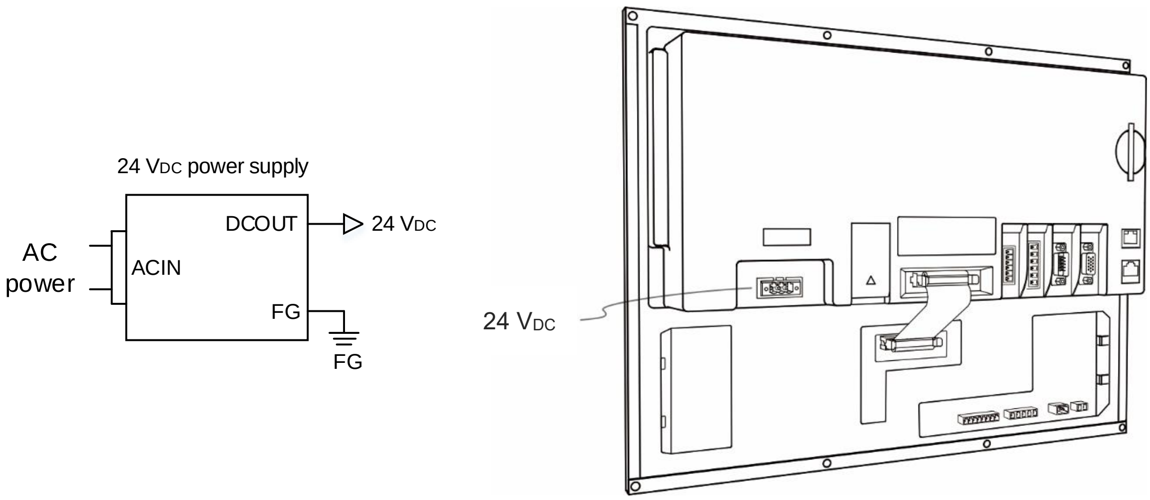
<table>
<tr><th>Symbol</th><th>Function</th><th colspan="2">Description</th></tr>
<tr><td rowspan="8">REMOTE I/O</td><td rowspan="8">Remote I/O module connector</td><td colspan="2">Remote I/O module communication signals, including X256 - X511 and Y256 - Y511. Each module has 32 DI points and 32 DO points, and the controller can connect to up to 8 modules.</td></tr>
<tr><td>Pin No.</td><td>Description</td></tr>
<tr><td>P1</td><td>TX+</td></tr>
<tr><td>P2</td><td>TX-</td></tr>
<tr><td>P3</td><td>RX-</td></tr>
<tr><td>P4</td><td>RX+</td></tr>
<tr><td>P5</td><td>GND</td></tr>
<tr><td>P6</td><td>SHIELD</td></tr>
<tr><td>PAN2</td><td>Connector for machine operation panel B</td><td colspan="2">The USB host connector for connecting to machine operation panel B. No connection to USB devices.</td></tr>
<tr><td>USB</td><td>USB connector</td><td colspan="2">USB connectors for connecting to USB devices, such as flash drive, mouse, and keyboard. Connection in series is not supported by this connector.</td></tr>
<tr><td>VGA</td><td>VGA connector</td><td colspan="2">Connect to an external monitor (only monitors with the refresh rate of 60 Hz is supported).</td></tr>
<tr><td rowspan="10">ETHERNET</td><td rowspan="10">Ethernet connector</td><td colspan="2">Connects to PC with an RJ45 connector and a network cable.
Pin definition of both ends:</td></tr>
<tr><td>Pin No. / color of end A</td><td>Pin No. / color of end B</td></tr>
<tr><td>1: orange & white</td><td>1: orange & white</td></tr>
<tr><td>2: orange</td><td>2: orange</td></tr>
<tr><td>3: green & white</td><td>3: green & white</td></tr>
<tr><td>4: blue</td><td>4: blue</td></tr>
<tr><td>5: blue & white</td><td>5: blue & white</td></tr>
<tr><td>6: green</td><td>6: green</td></tr>
<tr><td>7: brown & white</td><td>7: brown & white</td></tr>
<tr><td>8: brown</td><td>8: brown</td></tr>
<tr><td>DMCNET</td><td>DMCNET connector</td><td colspan="2">Connect to Delta's DMCNET servo drive with standard RJ45 connector. Wiring method is the same as that of the Ethernet connector.</td></tr>
<tr><td>MODE</td><td>Debugging mode</td><td colspan="2">Debugging mode switch</td></tr>
</table>

3

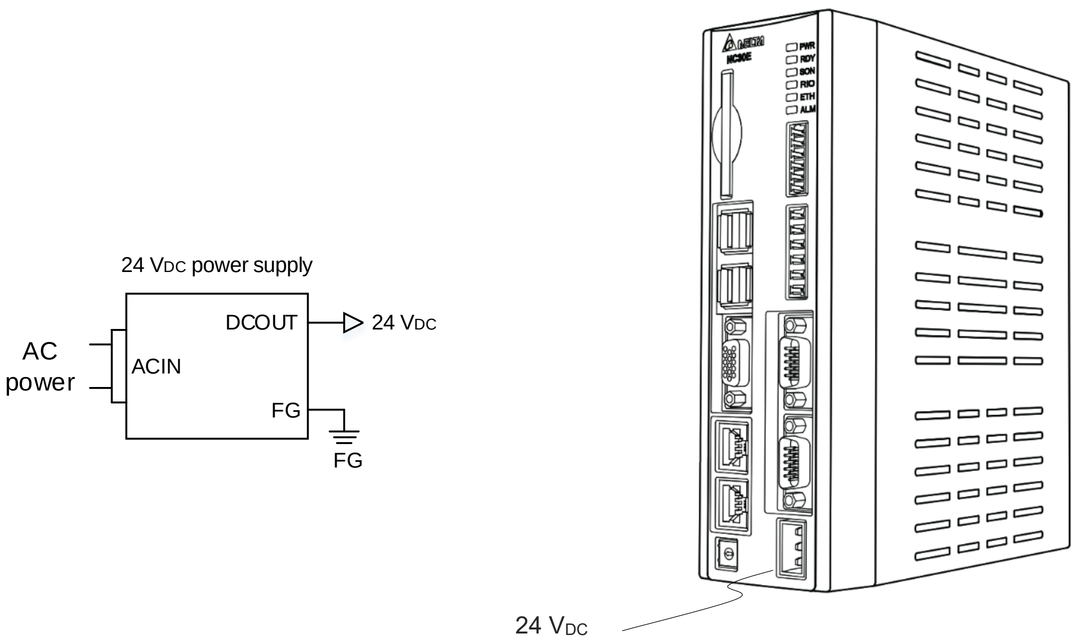
3.3 Wiring for power connector

The wiring method for the power connector of the NC series controller is apply power directly to the 24 V_{DC} connector.

- NC2 series models



- OPENCNC series models

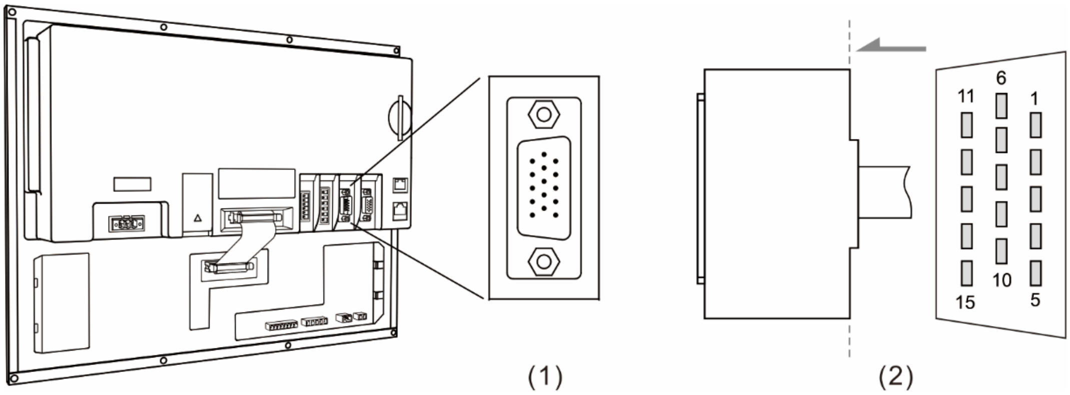


3

3.4 Wiring for RS-485 connector

The NC series controller has one RS-485 connector for serial communication with external devices.

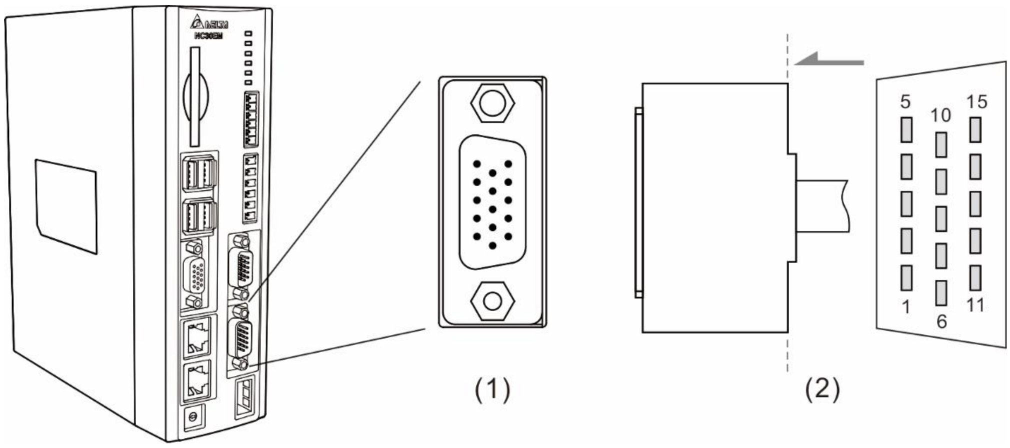
The pin assignment for NC2 series models is as follows.



(1) RS-485 connector (female); (2) RS-485 connector (male)

Model	Symbol	Pin No.	Function description
NC2_ _	SPINDLE	2	D+
		3	D-

The pin assignment for OPENCNC series models is as follows.



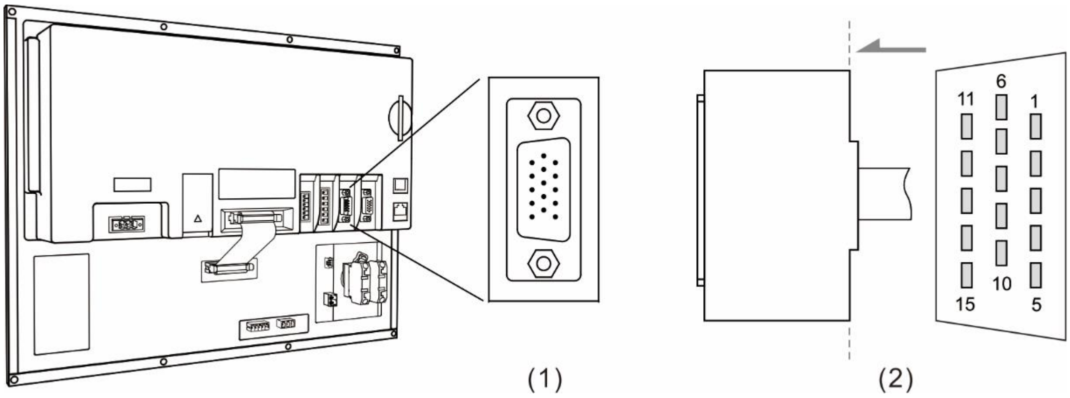
(1) RS-485 connector (female); (2) RS-485 connector (male)

Model	Symbol	Pin No.	Function description
NC30E_	SPINDLE	2	D+
		3	D-

3.5 Wiring for spindle feedback input connector

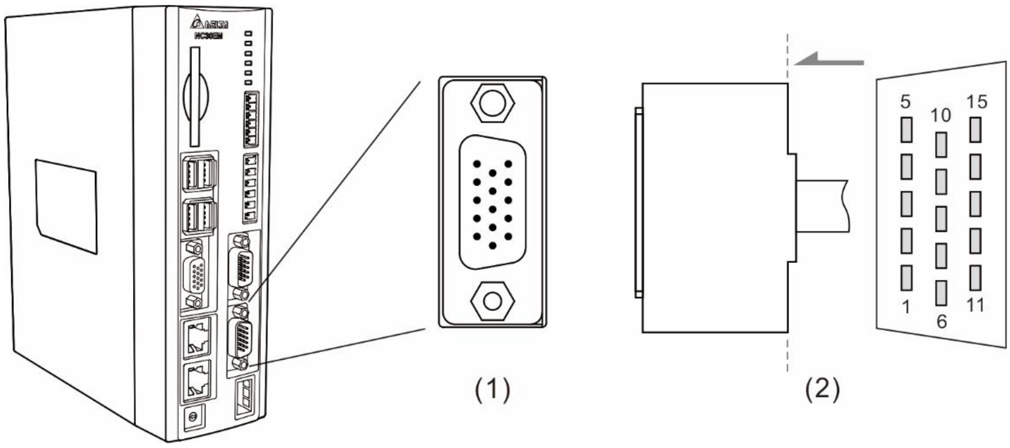
The NC series controller has one set of spindle feedback input.

The pin assignment for NC2 series models is as follows.



(1) Spindle feedback input connector (female); (2) Spindle feedback input connector (male)

The pin assignment for OPENCNC series models is as follows.



(1) Spindle feedback input connector (female); (2) Spindle feedback input connector (male)

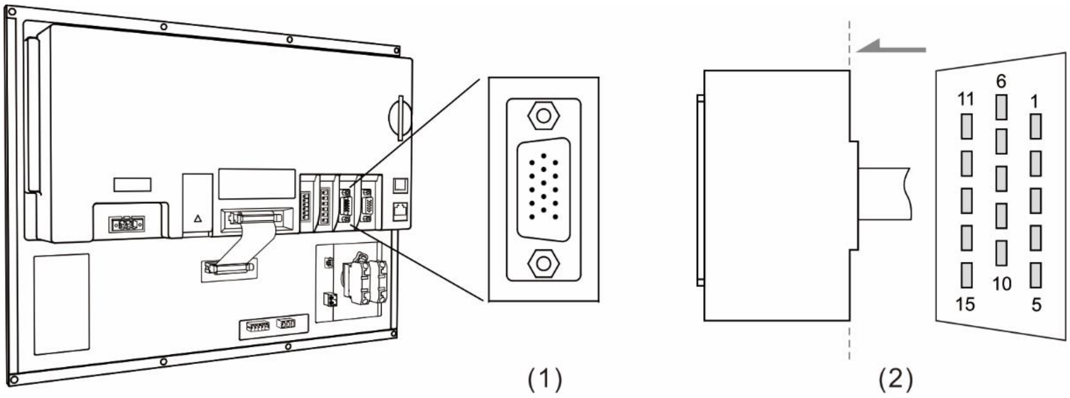
The function description of each pin is as follows.

Model	Symbol	Pin No.	Function description	
All	SPINDLE	8	SP_A+	Spindle encoder A+ pulse input
		9	SP_A-	Spindle encoder A- pulse input
		10	DC +5V_OUT	Spindle encoder power output (+5 V_{DC})
		11	SP_B+	Spindle encoder B+ pulse input
		12	SP_B-	Spindle encoder B- pulse input
		13	SP_Z+	Spindle encoder Z+ pulse input
		14	SP_Z-	Spindle encoder Z- pulse input
		15	GND	Ground for spindle encoder power

3.6 Wiring for spindle analog output connector

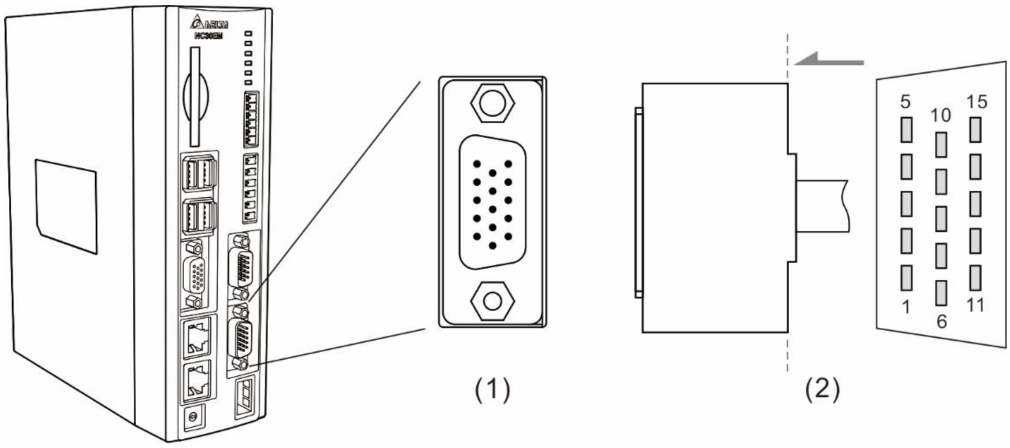
The NC series controller has one set of spindle analog output for controlling the spindle speed.

The pin assignment for NC2 series models is as follows.



(1) Spindle analog output connector (female); (2) Spindle analog output connector (male)

The pin assignment for OPENCNC series models is as follows.



(1) Spindle analog output connector (female); (2) Spindle analog output connector (male)

The function description of each pin is as follows.

Model	Symbol	Pin No.	Function description	
All	SPINDLE	4	SP_OUT	Spindle analog output
		5	SP_GND	Ground for spindle analog output signal

Settings for analog spindle:

Step1: set Pr399.

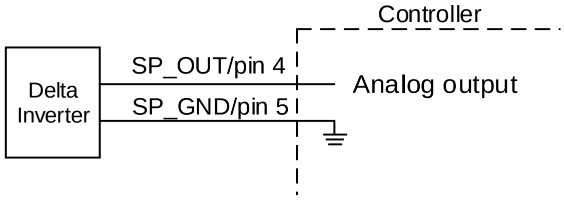
Bit	Description	Setting range
0	Spindle function 0: off; 1: on	0 - 1
1	Closed-loop control flag 0: off; 1: on (feedback encoder is required)	0 - 1
2 - 3	Spindle output mode 0: DMCNET (servo spindle); 1: reserved; 2: EDAC (analog output)	0 - 2

Bit	Description	Setting range
4	Speed control mode 1: PUU	1
5	Spindle encoder magnification 0: 1000 times; 1: 4 times	0 - 1

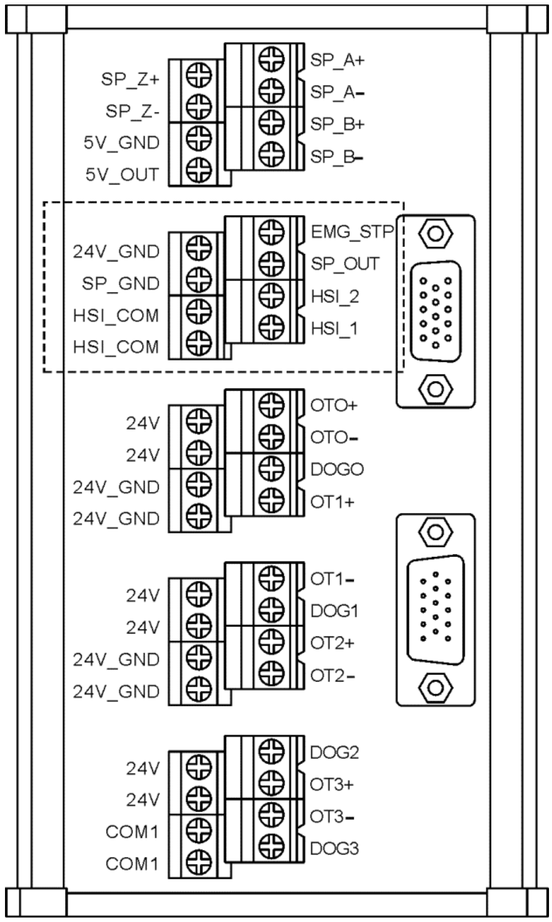
3

Step 2: in the [CONFIG] (channel setting) screen, enable SP1 and set its port number to 10.

Step 3: when using analog spindle to output signals, connect to pins 4 and 5 of the SPINDLE connector. The outputs 0 - 10 V_{DC} correspond to S0 (zero speed) to maximum speed. (The resolution of -10V to +10V is 12-bit.)



Inverter card (NC-EXM_S01)

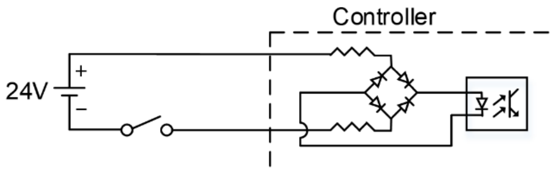


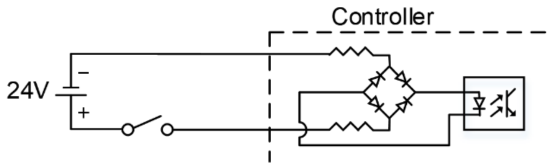
3.7 Wiring for HSI connector

The NC series controller has two sets of hardware high-speed counter signal input. For the wiring of high-speed counter input (bi-directional coupling), the maximum input bandwidth is up to 5 MHz; voltage: 22 - 26V; permissible current: 8 - 20 mA; surge current: below 50 mA.
The corresponding special M relay for HSI_1 is M2142, which is also applicable to G31 Skip command.

Settings of relevant parameters.

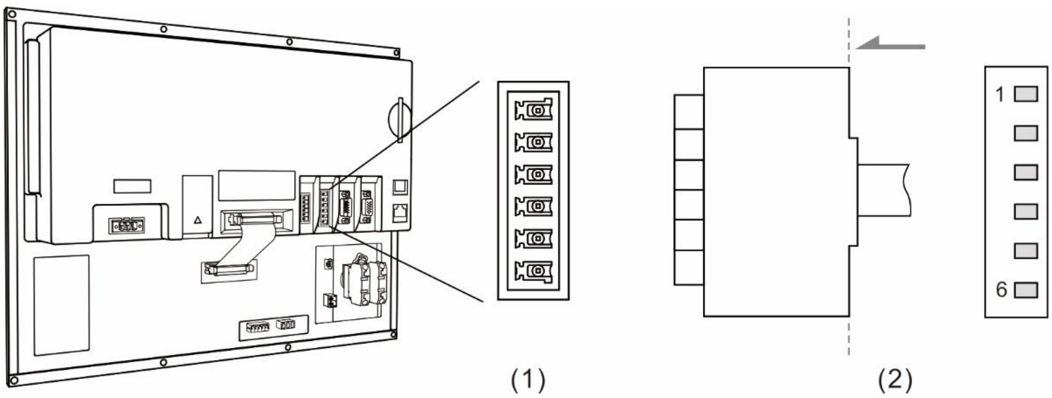
Parameter address		Parameter name	Function
Pr25	Bit 0	G31 high speed input 1 contact	0: NC; 1: NO
	Bit 1	G31 high speed input 2 contact	0: NC; 1: NO
Pr46	Bit 4	G31 high speed input 1 switch	0: off; 1: on
	Bit 5	G31 high speed input 2 switch	0: off; 1: on
Pr307	Bit 4 - 5	G31 input source	0: off; 1: HSI 1; 2: HSI 2; 3: HSI 1 & 2





Note: the connection direction of the external power input for HSI does not affect the operation.

The pin assignment for NC2 series models is as follows.



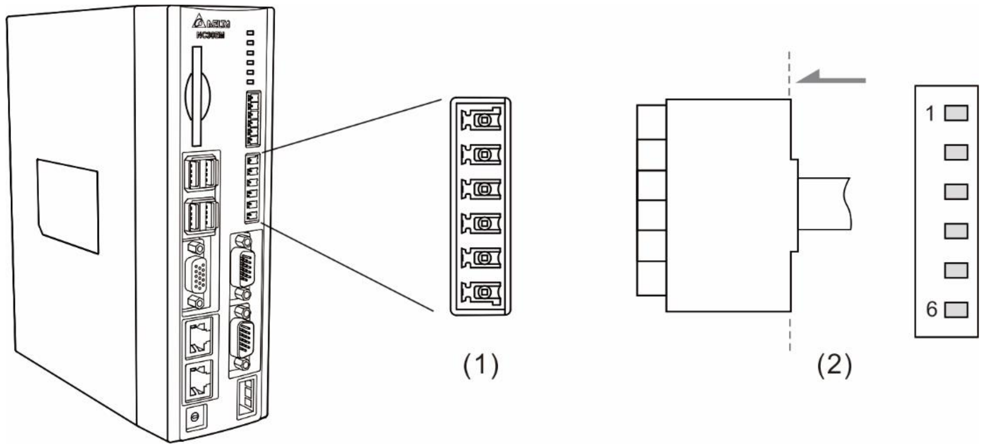
(1) HSI connector (female); (2) HSI connector (male)

The function description of each pin is as follows.

Model	Symbol	Pin No.	Function description	
NC2_ _	HSI	3	HSI_1	High-speed counter input 1 (10 mA)
		4	HSI_COM	High-speed counter COM; connects to +24 V_{DC} or 0V
		5	HSI_2	High-speed counter input 2 (10 mA)
		6	HSI_COM	High-speed counter COM; short-circuit P4 and P6.

3

The pin assignment for OPENCNC series models is as follows.



(1) HSI connector (female); (2) HSI connector (male)

The function description of each pin is as follows.

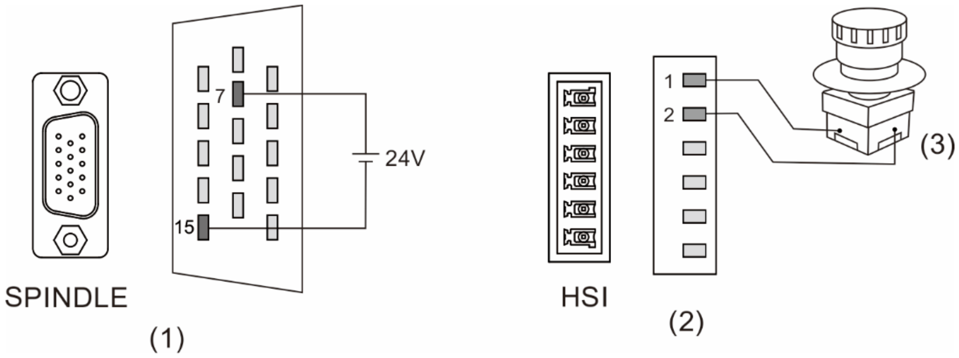
Model	Symbol	Pin No.	Function description	
NC30E_	HSI	3	HSI_1	High-speed counter input 1 (10 mA)
		4	GND	GND
		5	HSI_2	High-speed counter input 2 (10 mA)
		6	HSI_COM	High-speed counter COM; connects to +24 V_{DC} or 0V

Note: HSI_1 and HSI_2 can output +5V power, so both of them can form a circuit with GND.

3.8 Wiring for emergency stop

The NC series controller has one set of emergency stop signal input.

The pin assignment for NC2 series models is as follows.

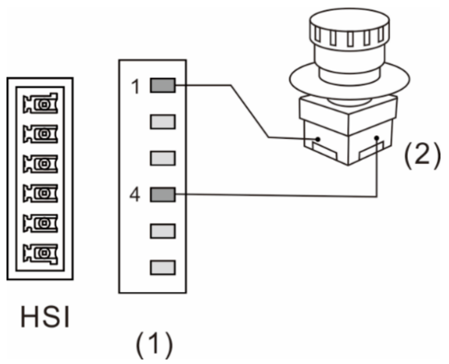


(1) & (2) EMG pins; (3) Emergency stop button

The function description of each pin is as follows.

Model	Symbol	Pin No.	Function description	
NC2__	SPINDLE	7	EMG_IN	Emergency stop power input (+24 V_{DC})
		15	GND	Ground for emergency stop power input
	HSI	1	EMG_IN	Emergency stop input
		2	EMG_GND	Ground for emergency stop input

The pin assignment for OPENCNC series models is as follows.



(1) EMG pins; (2) Emergency stop button

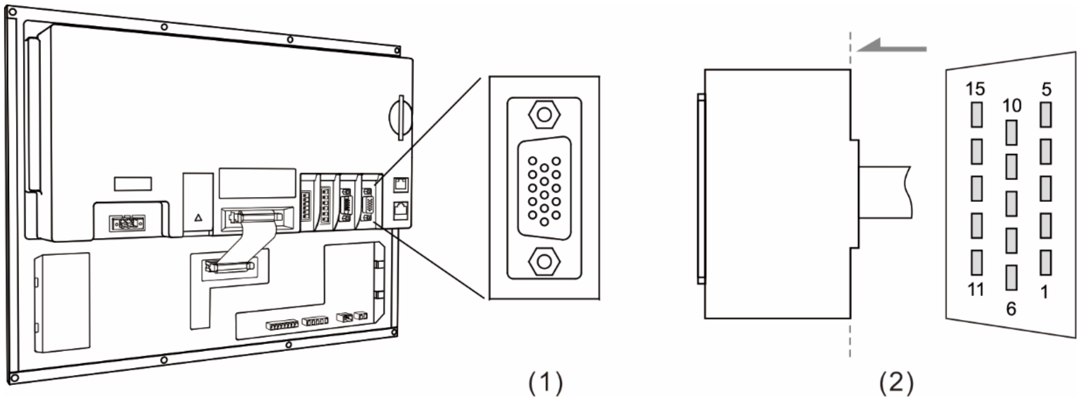
The function description of each pin is as follows.

Model	Symbol	Pin No.	Function description	
NC30E_	HSI	1	EMG_IN	EMG (+5 V_{DC} output)
		4	GND	GND

3.9 Wiring for MPG connector

The NC series controller has one MPG connector for receiving MPG pulses. This connector supplies +5 V_{DC} power which directly supplies power to the MPG.

The pin assignment for NC2 series models is as follows.

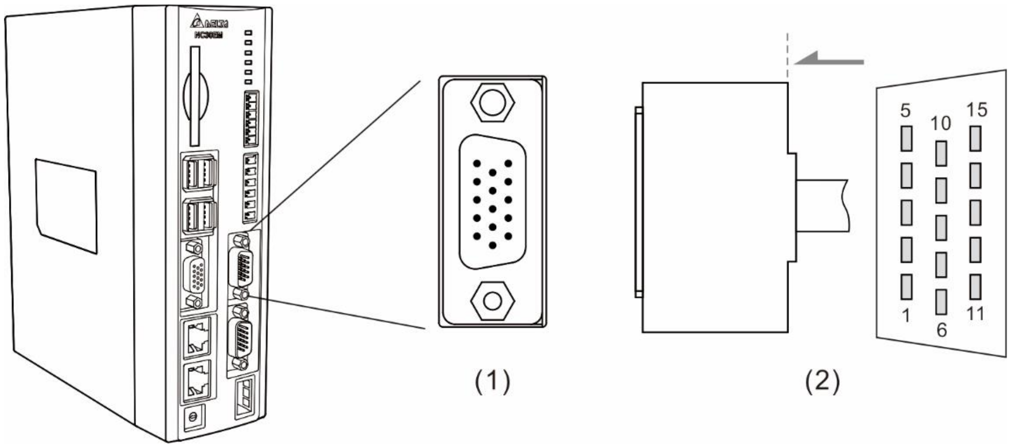


(1) MPG connector (female); (2) MPG connector (male)

The function description of each pin is as follows.

Model	Symbol	Pin No.	Function description	
NC2_ _	MPG	1	DI_COM	DI_COM; connects to +24 V_{DC} or 0V
		2	DI_1	X28
		3	DI_2	X29
		4	DI_3	X30
		5	DI_4	X31
		6	DI_5	X32
		7	DI_6	X33
		8	DI_7	X26
		9	DO_8	Y27
		10	DC +5V_OUT	DC +5V_OUT
		11	XA+	XA+
		12	XA-	XA-
		13	XB+	XB+
		14	XB-	XB-
		15	GND	GND

The pin assignment for OPENCNC series models is as follows.

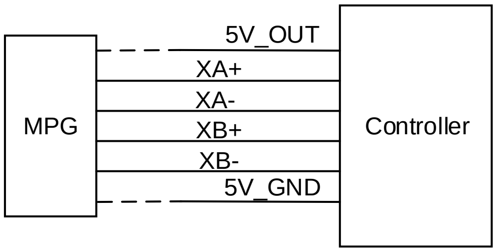


(1) MPG connector (female); (2) MPG connector (male)

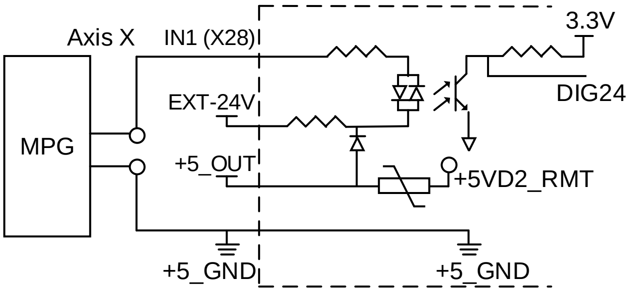
The function description of each pin is as follows.

Model	Symbol	Pin No.	Function description	
NC30E_	MPG	1	DI_COM	DI_COM; connects to +24 V_{DC} or 0V
		2	DI_1	X28
		3	DI_2	X29
		4	DI_3	X30
		5	DI_4	X31
		6	DI_5	X32
		7	DI_6	X33
		8	DI_7	X34
		9	DI_8	X35
		10	DC +5V_OUT	DC +5V_OUT
		11	XA+	XA+
		12	XA-	XA-
		13	XB+	XB+
		14	XB-	XB-
		15	GND	GND

Wiring for MPG pulse input. The controller supplies 5 V_{DC} power to the MPG.

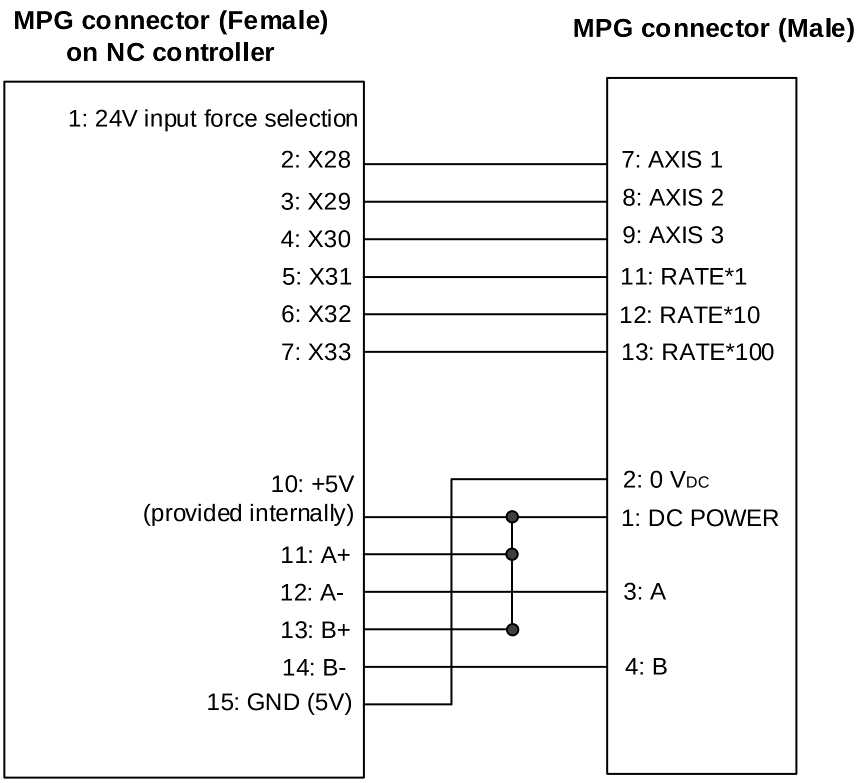


DI pin wiring:

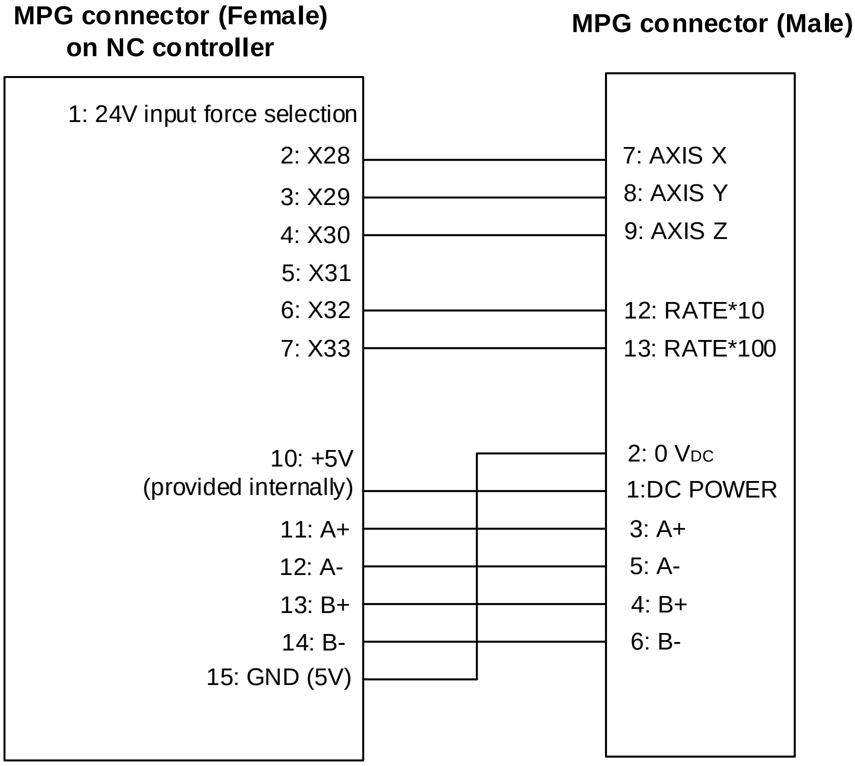


The signal type determines the wiring for MPG (only applicable to 100-ppr type), including single-ended type (EHDW-BA6SI) and differential type (EHDW-BE6SI).

Wiring diagram for single-ended type MPG (EHDW-BA6SI):

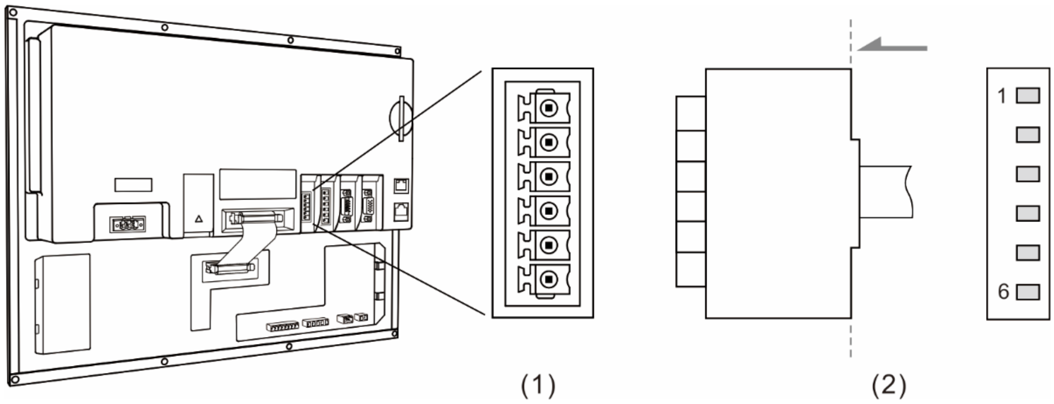


Wiring diagram for differential type MPG (EHDW-BE6SI):



3.10 Wiring for Remote I/O connector

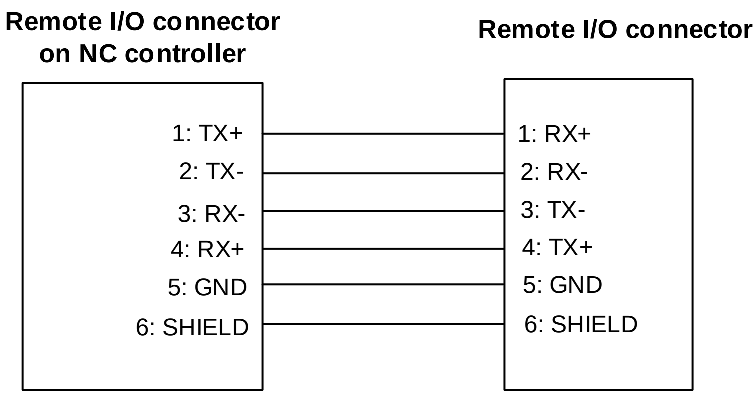
The NC series controller has a remote I/O connector which can connect to up to 8 extension modules with 256 DI and 256 DO points. The pin assignment is as follows.



(1) Remote I/O connector (female); (2) Remote I/O connector (male)

Pin No.	Function description
1	TX+
2	TX-
3	RX-
4	RX+
5	GND
6	SHIELD

Wiring diagram for remote I/O connector:



There are two types of remote I/O module: opto-isolated type (NC-EIO-T3232) and relay type (NC-EIO-R3216 / NC-EIO-R2010).

3

1. Opto-isolated type: NC-EIO-T3232

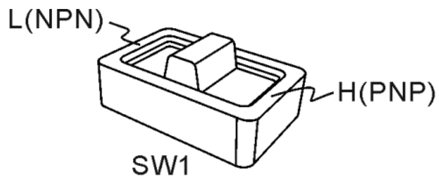
The opto-isolated type remote I/O module can connect to the NC controller using the RS-422 communication. Station numbers can be selected on the board and there is an offset of 32 points for the I/O address of every additional station. Thus, the first station starts from X256 / Y256, the second station starts from X288 / Y288, and so on. The controller can connect to up to 8 modules with 256 DI and 256 DO points available.

2. Relay type: NC-EIO-R3216 / NC-EIO-R2010

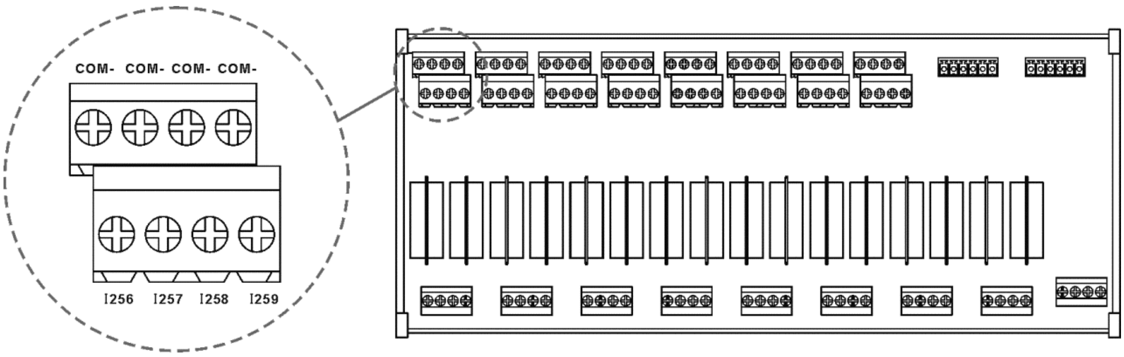
The relay type remote I/O module can connect to the NC controller using the RS-422 communication. Station numbers can be selected on the board and there is an offset of 32 points for the I/O address of every additional station. Thus, the first station starts from X256 / Y256 and the second station starts from X288 / Y288. This module supports 32 DI and 16 DO points, so the rest of the 16 DO points cannot be used, but the next station still has an offset of 32 points for the Y address.

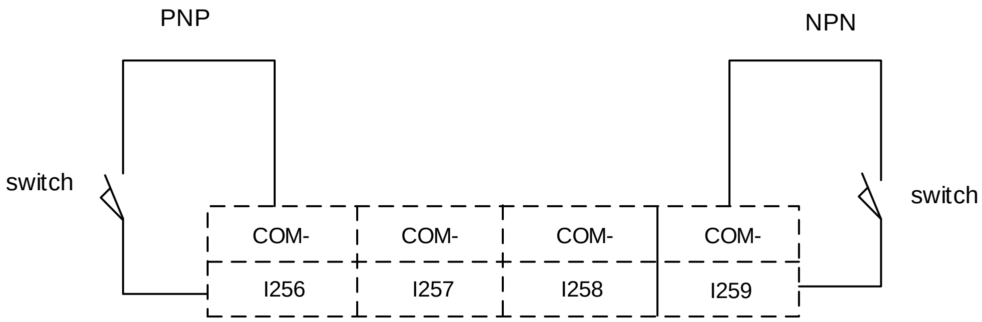
Wiring of remote I/O module:

COM- is for signal current and is prohibited from connecting to 24 V_{DC} or 0V power. Switch to H (PNP) or L (NPN) to select the signal type.

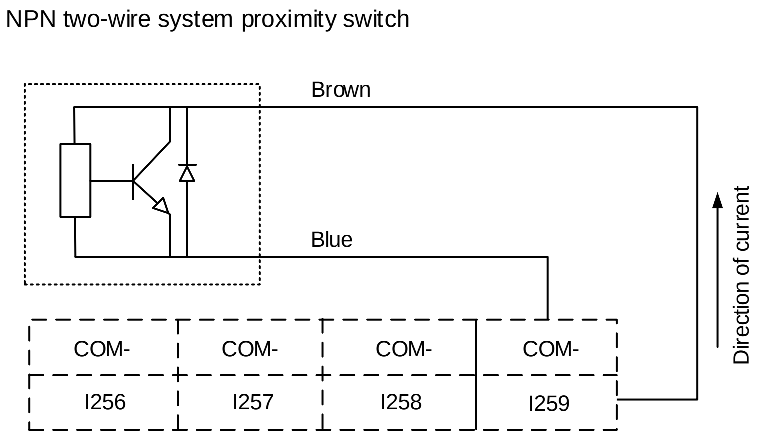


Wiring example for the button and mechanical switch:

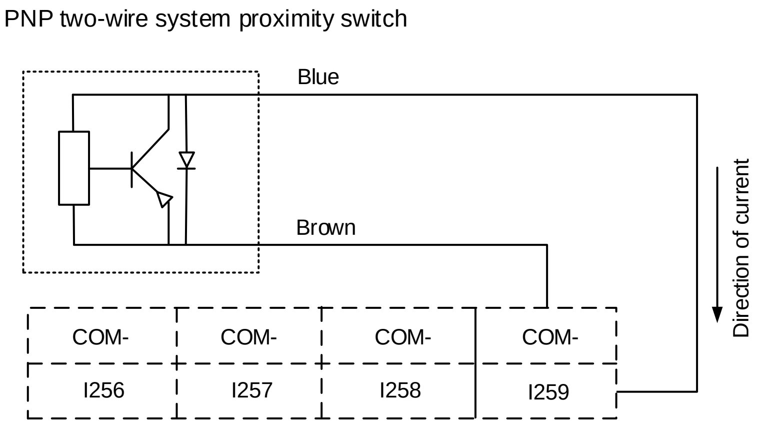




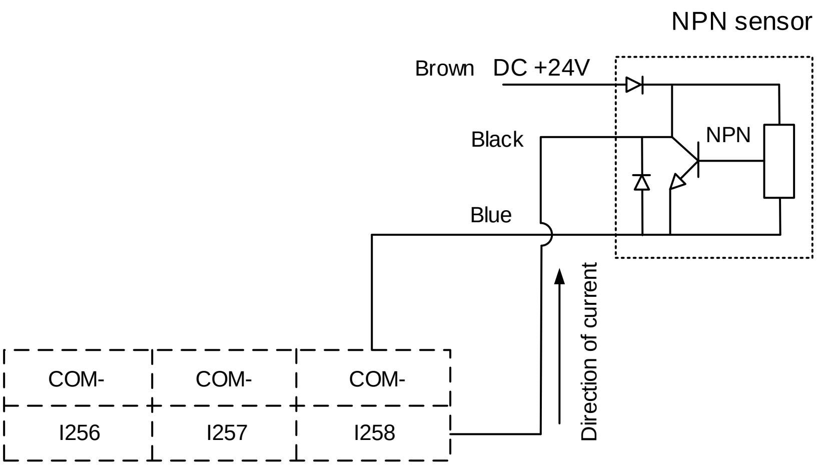
Wiring example for the proximity switch of NPN two-wire system:



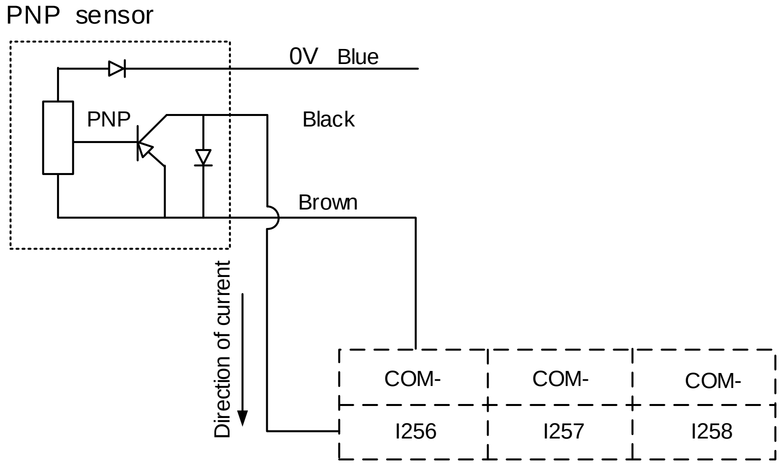
Wiring example for the proximity switch of PNP two-wire system:



Wring example for the proximity switch of NPN three-wire system:



Wring example for the proximity switch of PNP three-wire system:

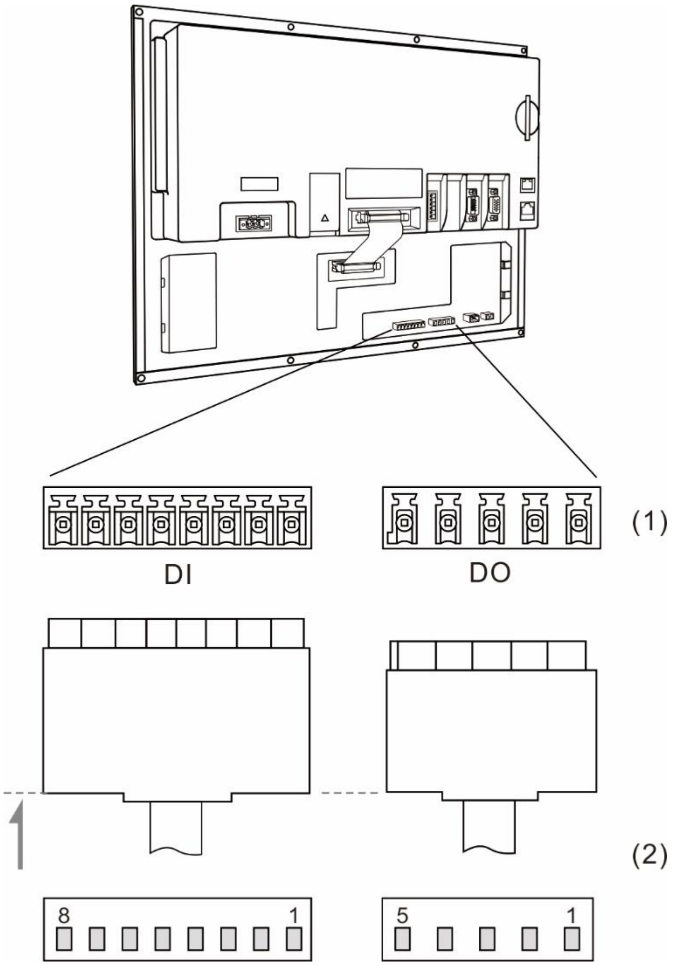


3

3.11 Wiring for local I/O connector

The pin assignment for the local I/O connector of the NC200 series controllers is as follows.

Note: NC30E(H) series models do not have the local I/O connector.

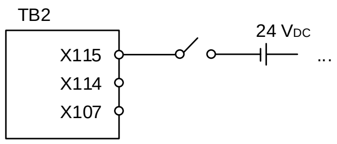


(1) Local I/O connector (female); (2) Local I/O connector (male)

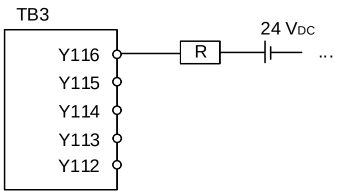
DI			
Pin 1	X112	Pin 2	X113
Pin 3	X114	Pin 4	X115
Pin 5	X116	Pin 6	X117
Pin 7	X118	Pin 8	X119

DO			
Pin 1	Y112	Pin 2	Y113
Pin 3	Y114	Pin 4	Y115
Pin 5	Y116	-	-

DI wiring; connects to an external power supply.



DO wiring; connects to an external power supply.



(This page is intentionally left blank.)

Table of Group Function 4

This chapter provides tables of group function for you to quickly view all of the functions of the NC series controller.

4

4.1 Tables of group function

Available in all modes

POS			
Layer 1	Layer 2	Layer 3	Layer 4
ABS	-	-	-
REL (the clear function of the corresponding axis is available only when you connect the axis)	CLR ALL	-	-
	CLR X	-	-
	CLR Y	-	-
	CLR Z	-	-
	CLR A	-	-
	CLR B	-	-
	CLR C	-	-
MECH	-	-	-

EDIT mode

PRG_File manage			
Layer 1	Layer 2	Layer 3	Layer 4
COPY F	-	-	-
PASTE	-	-	-
DEL (file / folder)	-	-	-
SEL TOGL / CANCEL	-	-	-
ALL SEL TOGL / CANCEL	-	-	-
File Manage	-	-	-
SORT	NAME	-	-
	SIZE	-	-
	DATE	-	-
NEW F	-	-	-
FOLDER	-	-	-
RENAME	-	-	-
FIND F	-	-	-
MERGE	-	-	-
MACRO	-	-	-
DXF	-	-	-
GRA EDT	-	-	-
INT MCR	-	-	-

EDIT mode

PRG_File edit			
Layer 1	Layer 2	Layer 3	Layer 4
COPY	-	-	-
PASTE	-	-	-
DEL	-	-	-
UNDO	-	-	-
B START	-	-	-
B END	-	-	-
LABLE	-	-	-
STRING	NEXT	-	-
	PREV	-	-
	REPLACE	-	-
	Replace all	-	-

AUTO mode

PRG			
Layer 1	Layer 2	Layer 3	Layer 4
SF SET	Spindle speed (S) range: 0 - Pr409 setting range Feedrate (F) range: 0 - Pr318 setting range		
SEARCH	RUN	-	-
BARCODE (set Pr10053 to enable this function)	LOAD	-	-
	CLR	-	-
	CLR ALL	-	-
FILE QUEUE	Set Pr10045 to enable or disable this function.		
File Manage	-	-	-

JOG / MPG mode

PRG			
Layer 1	Layer 2	Layer 3	Layer 4
SF SET	Spindle speed (S) range: 0 - Pr409 setting range Feedrate (F) range: 0 - Pr318 setting range		
TEACH (set Pr10044 to enable this function)	POSITION	-	-
	LINEAR	-	-
	CIRCULAR	P1	-
		P2	-
		P3	-
		PLANE	-
	DEL	-	-
	SAVE	-	-
	NEW FILE	-	-
	MECH / ABS	-	-

4

MDI mode

PRG			
Layer 1	Layer 2	Layer 3	Layer 4
LOAD	-	-	-
SAVE	-	-	-
CLR	-	-	-

HOME mode

PRG			
Layer 1	Layer 2	Layer 3	Layer 4
SF SET	Spindle speed (S) range: 0 - Pr409 setting range Feedrate (F) range: 0 - Pr318 setting range		

Available in all modes

OFS			
Layer 1	Layer 2	Layer 3	Layer 4
WEAR	ABS	-	-
	INC	-	-
	CLR ALL	-	-
	CLR ONE	-	-
	ABS / INC MODE	Set Pr10059 to enable this function (1: Abs; 2: Inc).	
OFFSET	ABS	-	-
	INC	-	-
	LEN OFST	-	-
	ABS OFST	-	-
	CLR ONE	-	-
	ABS / INC MODE	Set Pr10059 to enable this function (1: Abs; 2: Inc).	
MAGA	MAGA1 (functions in Layer 3 are only available in JOG mode)	SET	-
		RST ALL	-
		LOCK	-
		UNLOCK	-
	MAGA2 (functions in Layer 3 are only available in JOG mode)	SET	-
		RST ALL	-
		LOCK	-
		UNLOCK	-
MACRO (set Pr10045 to enable this function)	LOCAL	-	-
	GLOBAL	-	-
	HOLD	-	-
	EXTEND	-	-
	MECH	WR _ (axis coordinate)	-
	ABS	WR _ (axis coordinate)	-

<table>
<tr><th colspan="4">OFS</th></tr>
<tr><th>Layer 1</th><th>Layer 2</th><th>Layer 3</th><th>Layer 4</th></tr>
<tr><td rowspan="9">COORD</td><td rowspan="7">AUTO</td><td>CLR REL</td><td>-</td></tr>
<tr><td>CLR ALL</td><td>-</td></tr>
<tr><td>SET L</td><td>-</td></tr>
<tr><td rowspan="3">SET L/2</td><td>Point1</td></tr>
<tr><td>Point2</td></tr>
<tr><td>SET</td></tr>
<tr><td>SET P</td><td>-</td></tr>
<tr><td>ABS</td><td>-</td><td>-</td></tr>
<tr><td>INC</td><td>-</td><td>-</td></tr>
</table>

Available in all modes

<table>
<tr><th colspan="4">GRA</th></tr>
<tr><th>Layer 1</th><th>Layer 2</th><th>Layer 3</th><th>Layer 4</th></tr>
<tr><td>CENT SET</td><td>-</td><td>-</td><td>-</td></tr>
<tr><td>WIN RST</td><td>-</td><td>-</td><td>-</td></tr>
<tr><td rowspan="8">WIN SET</td><td>UP</td><td>-</td><td>-</td></tr>
<tr><td>DOWN</td><td>-</td><td>-</td></tr>
<tr><td>LEFT</td><td>-</td><td>-</td></tr>
<tr><td>RIGHT</td><td>-</td><td>-</td></tr>
<tr><td>ZM IN</td><td>-</td><td>-</td></tr>
<tr><td>ZM OUT</td><td>-</td><td>-</td></tr>
<tr><td>OK</td><td>-</td><td>-</td></tr>
<tr><td>CANCEL</td><td>-</td><td>-</td></tr>
<tr><td>PV(STEP)</td><td colspan="3">This function is only available in AUTO mode.</td></tr>
<tr><td>GRAPHIC</td><td>-</td><td>-</td><td>-</td></tr>
<tr><td>PREVIEW</td><td colspan="3">This function is only available in AUTO mode.</td></tr>
<tr><td>CLEAR</td><td>-</td><td>-</td><td>-</td></tr>
</table>

Available in all modes

ALM			
Layer 1	Layer 2	Layer 3	Layer 4
ALARM	-	-	-
HISTORY	CLR ALL	-	-

4

Available in all modes

DGN			
Layer 1	Layer 2	Layer 3	Layer 4
PROCESS	SET NR	-	-
	CLR TIME	-	-
	CLR NR	-	-
USR VAR	USR VAR	DEL	-
		US DEC	-
		HEX	-
		S DEC	-
		FLOAT	-
	SYS VAR	-	-
	M VAR	DEL	-
		US DEC	-
		HEX	-
		S DEC	-
		FLOAT	-
MLC	BIT	X	-
		Y	-
		M	-
		A	-
		T	-
		C	-
	REG	T	-
		C(16)	-
		C(32)	-
		D	-
		V	-
		Z	-
		US DEC	-
		HEX	-
		S DEC	-
		FLOAT	-
	DEV MON	US DEC	-
		HEX	-
		S DEC	-
		FLOAT	-

DGN			
Layer 1	Layer 2	Layer 3	Layer 4
MLC	EDITOR (only available in EDIT mode)	LD	-
		LDI	-
		LDP	-
		LDF	-
		OUT	-
		APP	-
		—	-
		|	-
		DEL V-LN	-
		ADD LN	-
		DEL LN	-
		DEL	-
		LABEL	-
		TABLE	-
		SYMBOL	X
			Y
			M
			A
			T
			C
			D
			P
			I
			DEL
			COPY
			PASTE
		SAVE	-
		IMPORT	IMPORT
		EXPORT	EXPORT
			NEW FILE
			NAME SR
			SIZE SR
			DATA SR
		JUMP TO	-
		SELECT	-
		CUT	-
		COPY	-
		PASTE	-
	SET (only available in EDIT mode)	ON	-
		OFF	-
		RUN / STP	-
	JUMP TO	-	-

DGN			
Layer 1	Layer 2	Layer 3	Layer 4
SYS MON	SRV MON	-	-
	I/O MON	-	-
	SYS VAR	-	
	VAR MON	SYS VAR	-
		CH VAR	-
		AXIS VAR	-
		IF VAR	-
		MLC VAR	-
		US DEC	-
		BIN	-
		HEX	-
		S DEC	-
STATUS	SYSTEM	-	-
	FW SN	-	-
	HW SN	-	-
	M INFO	DEL	-
PWD	S SCP	LOCK / UNLOCK	
		SYS CHECK	-
	M SCP	PWD CHG	-
		LOCK / UNLOCK	-
		RST U1	-
		RST U2	-
		FUN ENA	OK
			CANCEL
			DEFAULT
		RESET	-
	U1 SCP	PWD CHG	-
		LOCK / UNLOCK	-
	U2 SCP	PWD CHG	-
		LOCK / UNLOCK	-
	EXPIRE	SETTING	-
		RELEASE	-
		EXP SCP	PWD CHG
			LOCK / UNLOCK

<table>
<tr><th colspan="4">DGN</th></tr>
<tr><th>Layer 1</th><th>Layer 2</th><th>Layer 3</th><th>Layer 4</th></tr>
<tr><td rowspan="13">TUNING
(only available in JOG or MPG mode)
(set Pr10045 to enable or disable this function)</td><td>NEXT AX</td><td>-</td><td>-</td></tr>
<tr><td>READ</td><td>-</td><td>-</td></tr>
<tr><td>COMPUTE</td><td>-</td><td>-</td></tr>
<tr><td>WR GAIN</td><td>-</td><td>-</td></tr>
<tr><td>WR Notch</td><td>-</td><td>-</td></tr>
<tr><td>RUN</td><td>-</td><td>-</td></tr>
<tr><td>JOG←</td><td>-</td><td>-</td></tr>
<tr><td>JOG→</td><td>-</td><td>-</td></tr>
<tr><td>POS1</td><td>-</td><td>-</td></tr>
<tr><td>POS2</td><td>-</td><td>-</td></tr>
<tr><td rowspan="2">TAP RIV</td><td>TAP SET(1)</td><td rowspan="2">-</td></tr>
<tr><td>TAP SET(2)</td></tr>
<tr><td>SERVO</td><td>READ</td><td>-</td></tr>
<tr><td>TEXT WR</td><td colspan="3">Set Pr10045 to enable or disable this function.</td></tr>
<tr><td rowspan="3">IMPORT</td><td>IMPORT</td><td>-</td><td>-</td></tr>
<tr><td>SEL ALL</td><td>-</td><td>-</td></tr>
<tr><td>CLR ALL</td><td>-</td><td>-</td></tr>
<tr><td rowspan="3">EXPORT</td><td>EXPORT</td><td>-</td><td>-</td></tr>
<tr><td>SEL ALL</td><td>-</td><td>-</td></tr>
<tr><td>CLR ALL</td><td>-</td><td>-</td></tr>
<tr><td>LOGO WR</td><td colspan="3">Set Pr10045 to enable or disable this function.</td></tr>
</table>

4

4

Available in all modes

PAR			
Layer 1	Layer 2	Layer 3	Layer 4
PROCESS	-	-	-
OPERATE	-	-	-
MAGA	-	-	-
SPINDLE	-	-	-
MACHINE	-	-	-
HOME	-	-	-
ETH.	DEFAULT	-	-
COMP	OK	-	-
	um	-	-
	um+	-	-
	IMPORT	-	-
	IMPORT+	-	-
SYSTEM	DEFAULT	-	-
	COLOR	-	-
MLC	DEFAULT	-	-
	COLOR	-	-
GRAPHIC	DEFAULT	-	-
	COLOR	-	-
SERVO	READ	-	-
SEARCH	-	-	-
CONFIG (except AUTO and MDI modes)	OK	-	-
SET RIO (except AUTO and MDI modes)	OK	-	-
PAR GRP	SAVE	-	-
	DEL GRP	-	-
	WRT PAR	-	-
	RED PAR	-	-
	SRT PAR	-	-
	AVERAGE	-	-

Available in all modes

SOFT
You can configure the panel screen with the ScreenEditor software for application needs.

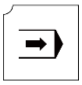
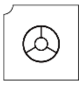
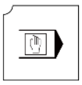
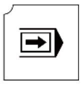
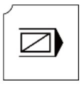
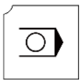
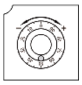
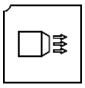
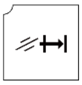
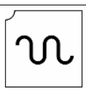
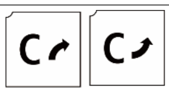
4.2 Machine operation panel A - function of keys

4

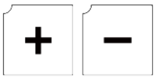
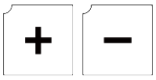
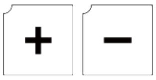
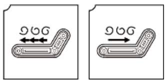
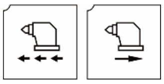
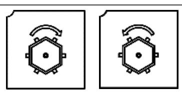
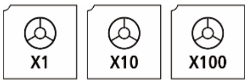
Symbol	Description	Supported mode / group
POS	The POS group key. Press to display the screen of coordinate setting.	All modes and groups
PRG	The PRG group key. Press to display the screen of program editing.	All modes and groups
OFS	The OFS group key. Press to display the screen of coordinate setting and tool offset setting.	All modes and groups
DGN	The DGN group key. Press to display the screen of diagnosis, system parameter, and system status.	All modes and groups
ALM	The ALM group key. Press to display the screen of alarm display.	All modes and groups
GRA	The GRA group key. Press to display the screen of graphic display.	All modes and groups
PAR	The PAR group key. Press to display the screen of parameter setting.	All modes and groups
SOFT	The SOFT group key. Press to display the configured panel screen.	All modes and groups
RESET	Reset key	All modes and groups
X I ~ D H	Axis position and command code keys	PRG
0 ^ ~ 9 /	Numeric keys (operation symbols)	PRG, OFS, DGN
• >	Decimal point key (operation symbol)	PRG, OFS
—]	Negative sign key (operation symbol)	PRG, OFS
▲ PAGE UP ▼ PAGE DN	PAGE UP and PAGE DN (page down) keys	PRG, OFS, DGN
⬆ ; ⬇ , ⬅ & ➡ #	Cursor keys (operation symbols)	PRG, OFS, DGN
HOME END	Home (end) key	PRG
SPACE	Space key	PRG

Symbol	Description	Supported mode / group
SHIFT	Shift key	PRG
DEL INS	Delete (insert) key	PRG
BACK SPACE	Back space key	PRG
ENTER	Enter key	PRG, OFS, DGN
EXIT	Exit key	PRG, DGN
( )	Parentheses key	PRG
◀ ▶	Left and right function keys	All modes and groups
F1 ~ F6	Function keys	All modes and groups

4.3 Machine operation panel B - function of keys

Symbol	Description
	Auto mode: for program execution
	Edit mode: for file management and program editing
	JOG mode: for manually operating the machine tool
	MPG mode: for operating the axis direction of the machine tool with MPG
	MDI mode: for simple program entering and execution
	Home mode: for rapid homing to the machine origin
	Single block execution: execute one single block at a time and then stop
	Single block skip: skip one single block when there is a “/ ” symbol in the block
	Optional stop: stop at the specified block if there is an M01 command in the block
	MPG simulation: during program execution, enable this function to control the execution speed with MPG
	Air blow switch
	Coolant switch: coolant ON / coolant OFF
	Limit cancellation: when the hardware limit is triggered, press this key to clear the alarm
	Light switch: light ON / light OFF
	Rapid traverse mode: execute axis movement based on the set rapid override
X ← X →	X axis direction: in JOG mode, manually move the X axis towards positive or negative direction
Y ↙ Y ↗	Y axis direction: in JOG mode, manually move the Y axis towards positive or negative direction
Z ↑ Z ↓	Z axis direction: in JOG mode, manually move the Z axis towards positive or negative direction
	Rotation axis direction: in JOG mode, manually run the rotation axis in forward or reverse direction

4

Symbol	Description
	Spindle forward: manually run the spindle in forward direction
	Spindle stop: manually stop the spindle rotation
	Spindle reverse: manually run the spindle in reverse direction
 Feedrate / JOG	Feedrate / JOG override: increment / decrement
 Rapid override	Rapid override: increment / decrement
 Spindle override	Spindle override: increment / decrement
	Chip discharge conveyor direction: motor runs in positive / negative direction
	Tailstock center: forward / backward
	Tool turret: clockwise / counterclockwise
	Spindle hydraulic chuck: release / tighten
 X1 X10 X100	MPG magnification selection: X1, X10, X100
CUM 1 CUM 2 CUM 3	User-defined keys

4.4 Table of corresponding buttons (for OPENCNC models)

NC key	PC keyboard	Description
F1 - F8 (function keys)	F1 - F8	Function keys
► (function key)	Tab	Next page (of the function bar)
◄ (function key)	Ctrl + Tab	Previous page (of the function bar)
POS	Ctrl + F1	The POS group key
PRG	Ctrl + F2	The PRG group key
OFS	Ctrl + F3	The OFS group key
DGN	Ctrl + F4	The DGN group key
ALM	Ctrl + F5	The ALM group key
GRA	Ctrl + F6	The GRA group key
PAR	Ctrl + F7	The PAR group key
SOFT	Ctrl + F8	The SOFT group key
Numeric keys	Numeric keys	-
Alphabetic keys	Alphabetic keys	-
Symbol keys	Symbol keys	-
Direction keys	Direction keys	-
PAGE UP / PAGE DN	Page Up / Page Down	-
BACK SPACE	Backspace	-
SPACE	Space	-
DEL / INS	Delete / Insert	-
SHIFT	Shift	-
HOME / END	Home / End	-
ENTER	Enter	-
EXIT	Esc	-
RESET	Ctrl + Esc	-
-	F12	Help (function descriptions for keys)
SHIFT + GRA	PrtScn	Screen capturing

(This page is intentionally left blank.)

Introduction to NC System Modes

5

This chapter introduces the seven system modes supported by the NC controller.

5.1 Auto mode (AUTO)

To execute a program, you have to open the file, switch the system to AUTO mode, and then press **CYCLE START**. In this mode, you can verify the machining program, cutting conditions, and position coordinates before execution as well as avoid unexpected execution by accidentally pressing **CYCLE START** in other modes. This mode is only for program execution rather than program editing or manual axis movement.

5.2 Program edit mode (EDIT)

You can edit a program in EDIT mode. In this mode, the editing functions in PRG group are enabled for you to edit the program. In addition, program execution and manual axis movement are not available in this mode.

5.3 Manual input mode (MDI)

In MDI mode, you can enter and execute a single block of program in the PRG group screen. In this mode, you can enter up to 14 program blocks in the PRG screen. General program editing, program execution, and manual axis operation are not available in this mode.

5.4 MPG mode (MPG)

In MPG mode, you can use the external MPG to manually operate the axes promptly and accurately. Program editing, program execution, and jog operation are not available in this mode.

5.5 Jog mode (JOG)

In JOG mode, press the axis direction keys on machine operation panel B to have the axes jog. Set the jog speed and moving distance with the JOG override key. You can move the work platform in high speed with the rapid traverse override key and axis direction keys. The axis moving speed is determined by the rapid override setting. Program execution and editing are not available in this mode.

5.6 Rapid mode (RAPID)

When in JOG mode, pressing **RAPID** during operation can switch the JOG speed to the set rapid traverse override.

5.7 Homing mode (HOME)

In HOME mode, you can return the axes to the machine origin by simply pressing the corresponding axis direction keys on machine operation panel B. After booting, you should set the system to HOME mode to have each axis return to the machine origin before executing the program. If you do not perform homing after booting, program execution is prohibited.

5.8 Group screen overview

A full range of information is provided on the screens of function groups of this controller. The following introduces some functions in the group screens.

■ **POS group**

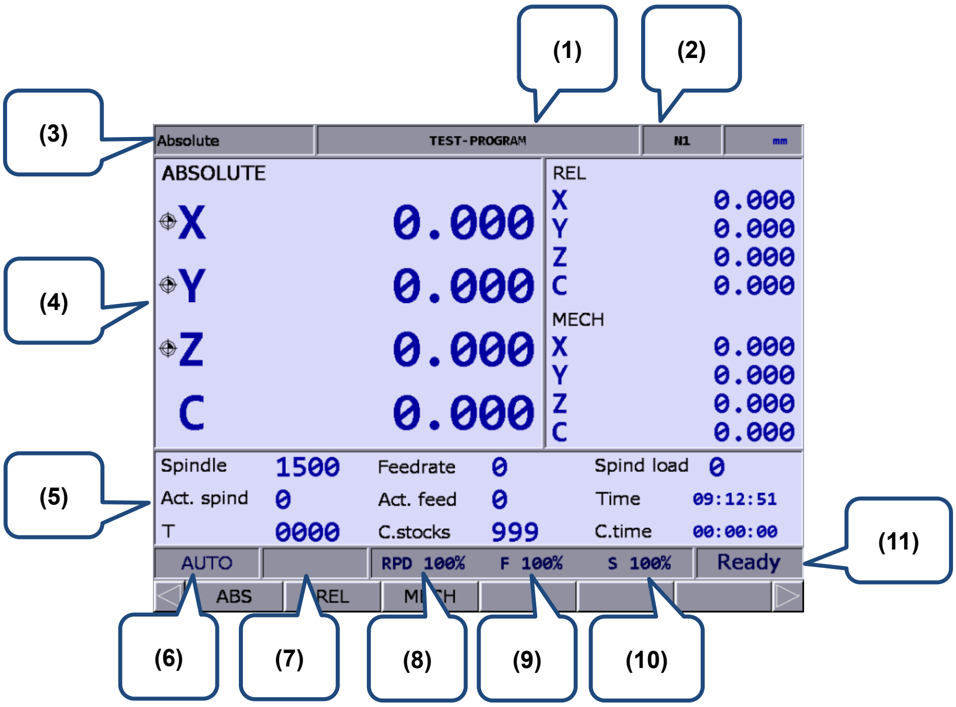


Figure 5.8.1

(1) Name of current program
(2) Currently executed program line
(3) Current group function
(4) Current coordinates
(5) Machining information:
Spindle speed: command value
Cutting feedrate: command value
Spindle load rate: %
Actual speed: rpm
Actual feedrate: mm/min
System time
Tool number (T)
Count of machining operations
Cutting time

(6) Current system mode
(7) Alarm display
(8) Rapid traverse override
(9) Feedrate override
(10) Spindle override
(11) System status

(11) displays the current status of the system for your reference. There are 7 system statuses with the display priority as follows: MLC stop > SV NO RDY (servo not ready) > Emg Stop (emergency stop) > PROC (in progress) > RUN (in execution) > STOP (program stops) > Ready.

■ **PRG group**

AUTO mode:

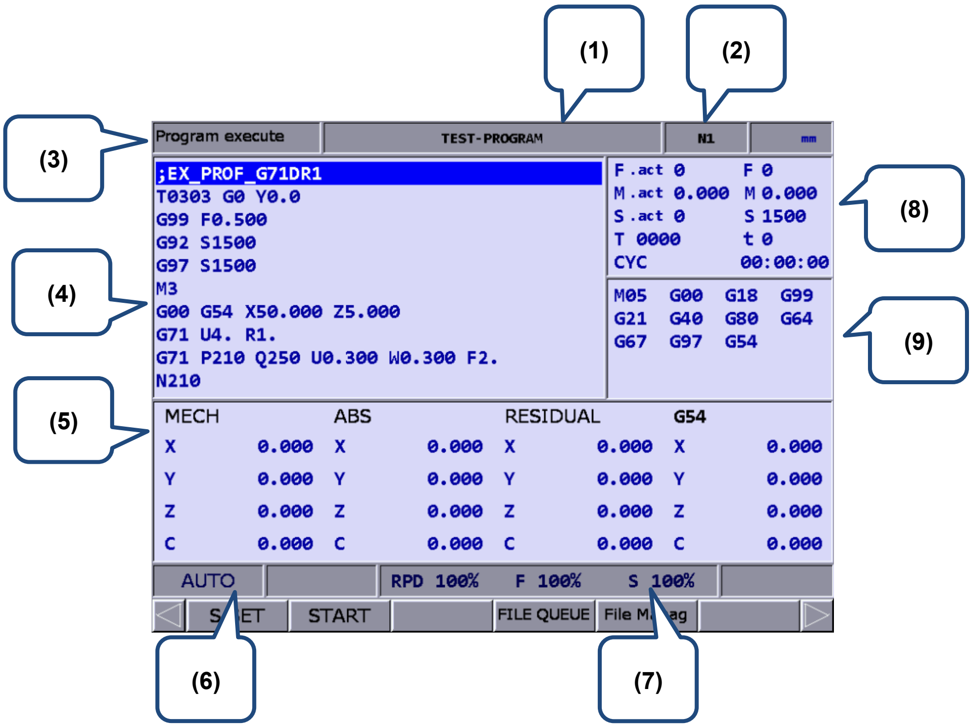


Figure 5.8.2

(1) Name of current program

(2) Currently executed program line

(3) Current group function

(4) Currently executed program content

(5) Coordinate information during program execution

(6) Current system mode

(7) Current override settings

(8) F.act: actual feedrate per minute

M.act: actual feedrate per revolution

S.act: actual spindle speed

T: tool number

F: feedrate (command value)

M: feedrate per revolution (command value)

S: spindle speed

t: dwelll time

CYC: cycle time

(9) Current command status

EDIT mode:

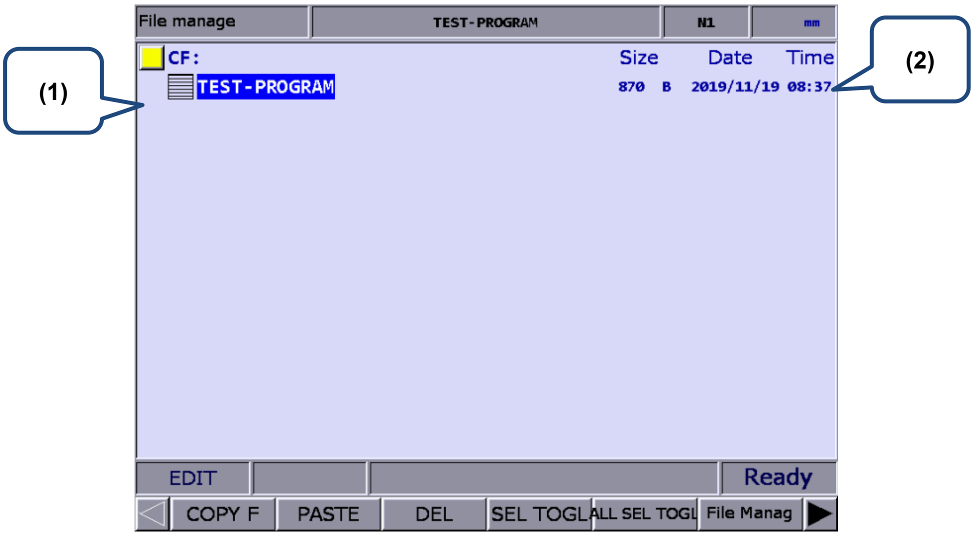


Figure 5.8.3

(1) File list: displays folders and program files

(2) File information: displays the size and modification date and time of the file or folder

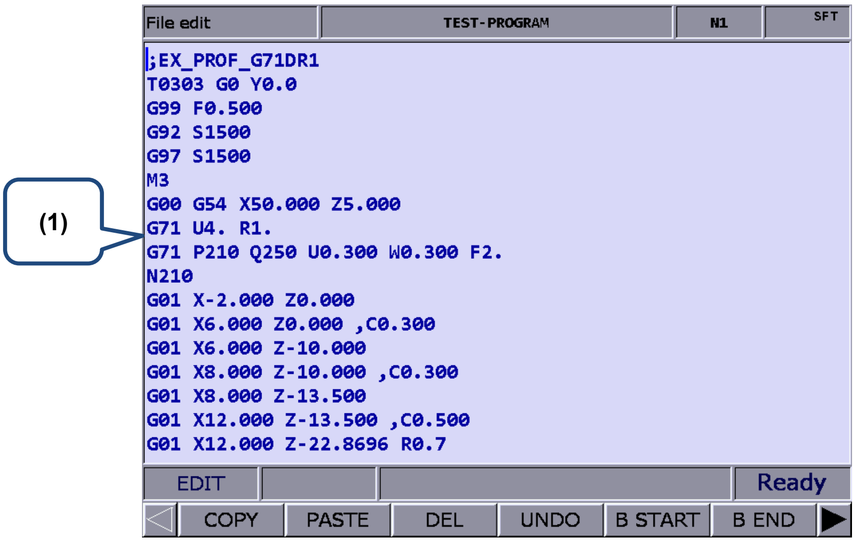


Figure 5.8.4

(1) File content: displays the content of the program file

MDI mode

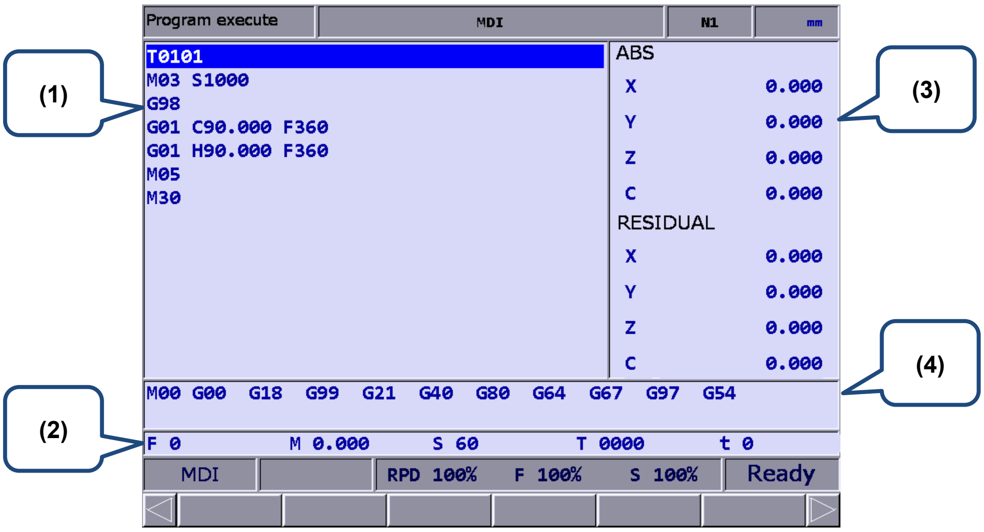


Figure 5.8.5

(1) MDI program

(2) Information of cutting feedrate, feedrate per revolution, spindle speed, tool number, and dwell time

(3) Coordinate information: absolute / residual coordinates

(4) Currently executed status commands

■ **OFS group**

Coordinate information:

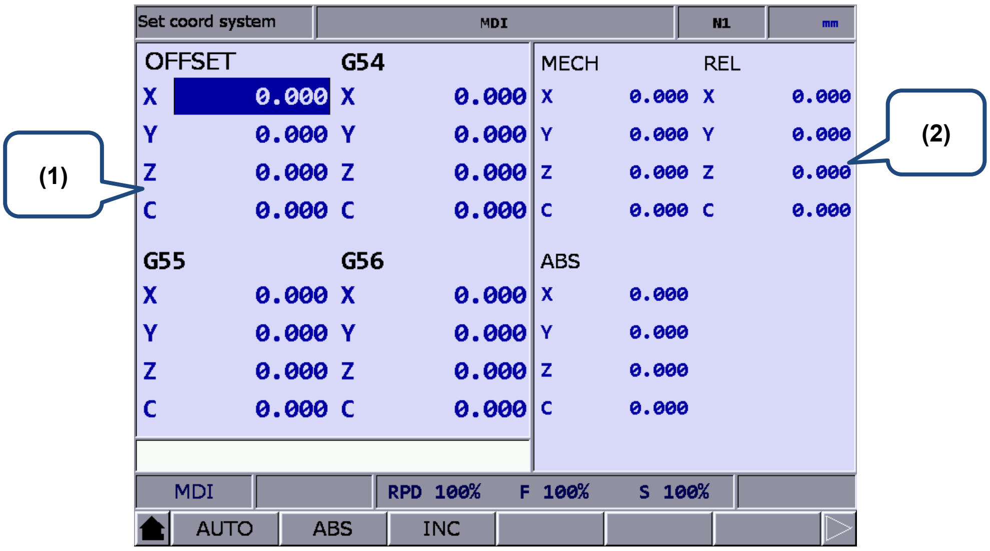


Figure 5.8.6

(1) Workpiece coordinate setting: offset coordinates, G54 - G59 coordinate system

(2) Coordinate information: machine / relative / absolute coordinates

Tool Wear:

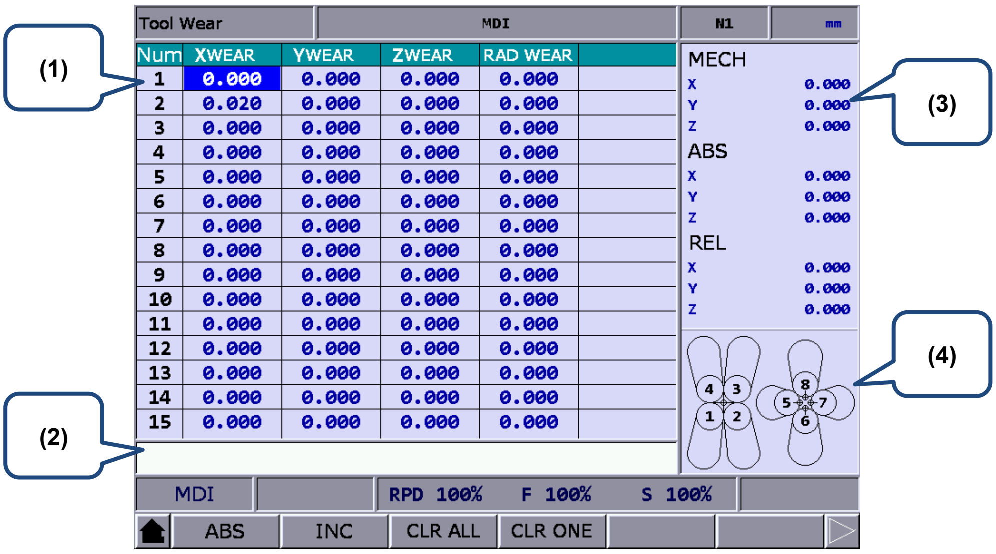


Figure 5.8.7

(1) Compensation number

(2) Input field for compensation data

(3) Auxiliary display: coordinate system

(4) Auxiliary display: illustration of tool tip type

Tool length (Tool Offset):

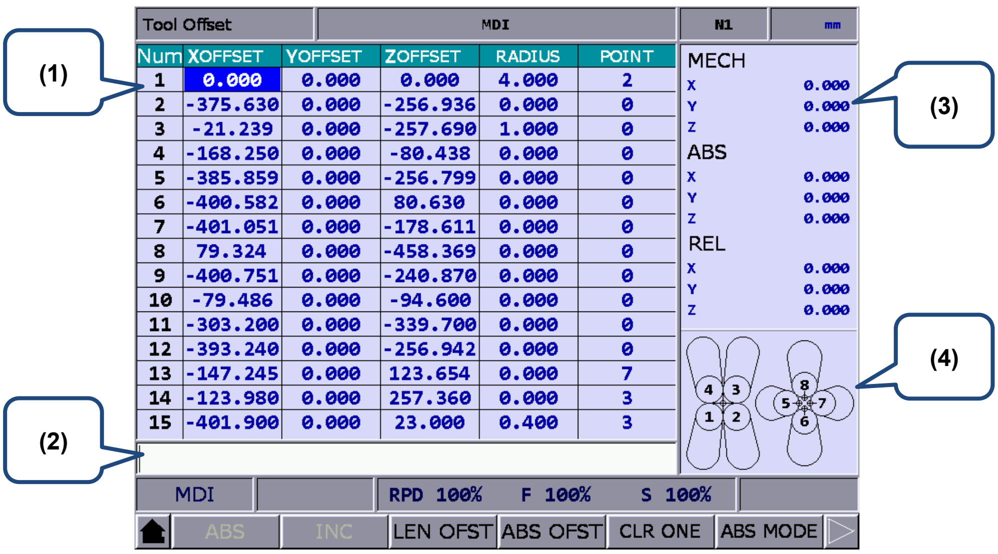


Figure 5.8.8

(1) Compensation number

(2) Input field for compensation data

(3) Auxiliary display: coordinate system

(4) Auxiliary display: illustration of tool tip type

■ DGN group

Servo Tuning:

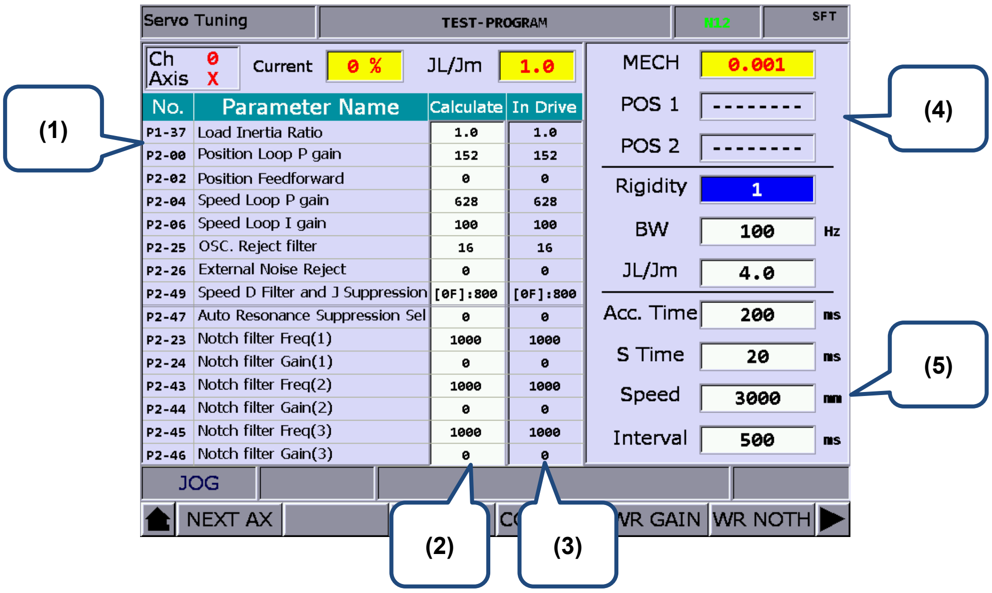


Figure 5.8.9

(1) Servo parameter: number and name of servo parameters

(2) Results after gain tuning: displays the calculation results of auto tuning

(3) System settings: displays the currently used servo settings

(4) Positioning setting: position 1 / position 2

(5) Tuning conditions

MLC Operation / Edit:

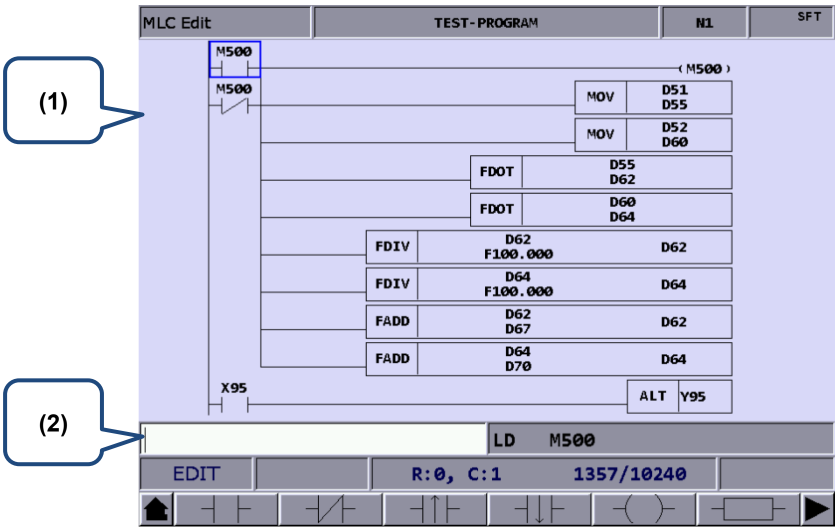


Figure 5.8.10

(1) MLC program

(2) Input field for command name

- **ALM group**

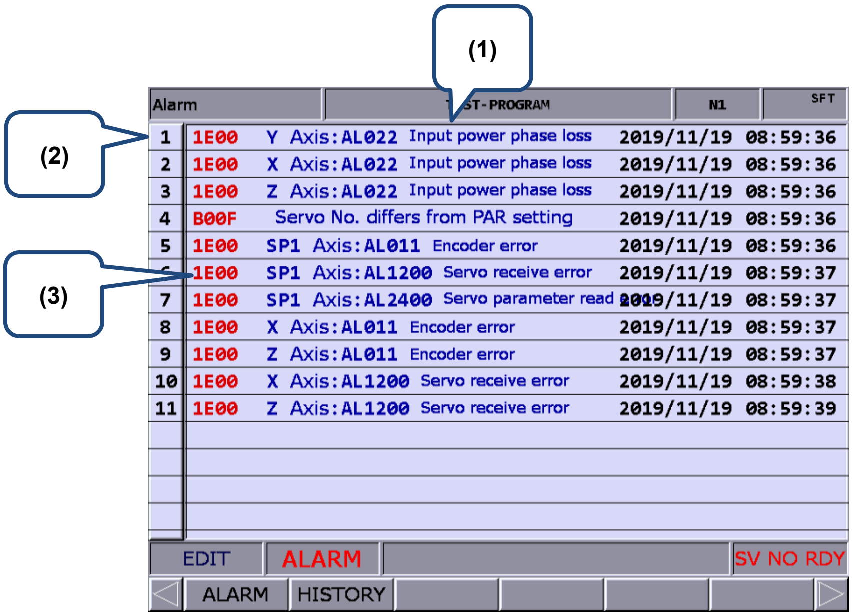


Figure 5.8.11

(1) Alarm message

(2) Sequence of alarm occurrence

(3) Alarm code

■ **GRA Group**

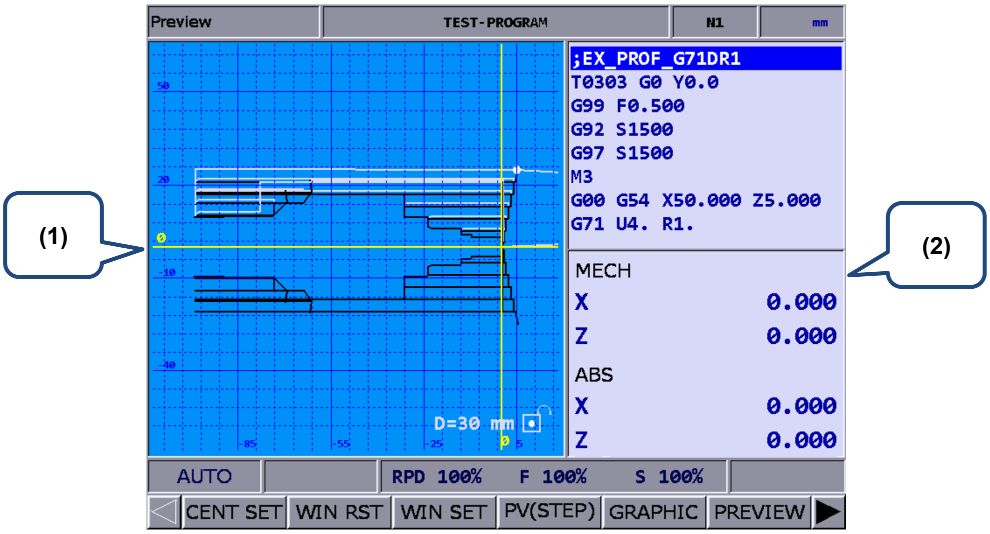


Figure 5.8.12 Screen of setting Pr14003 to 0

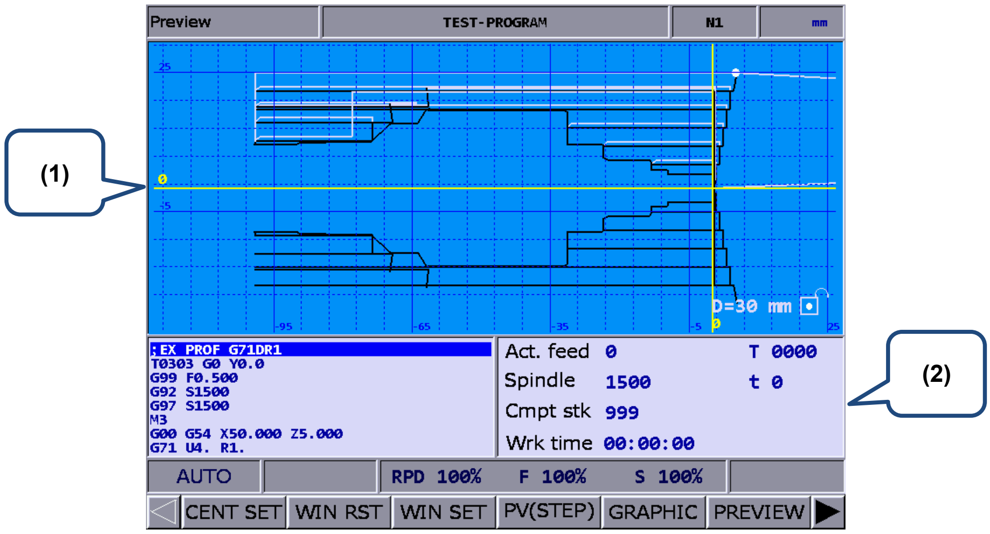


Figure 5.8.13 Screen of setting Pr14003 to 1

(1) Path diagram: displays the program path

(2) Displays the program in execution, system information, and coordinate information

(This page is intentionally left blank.)

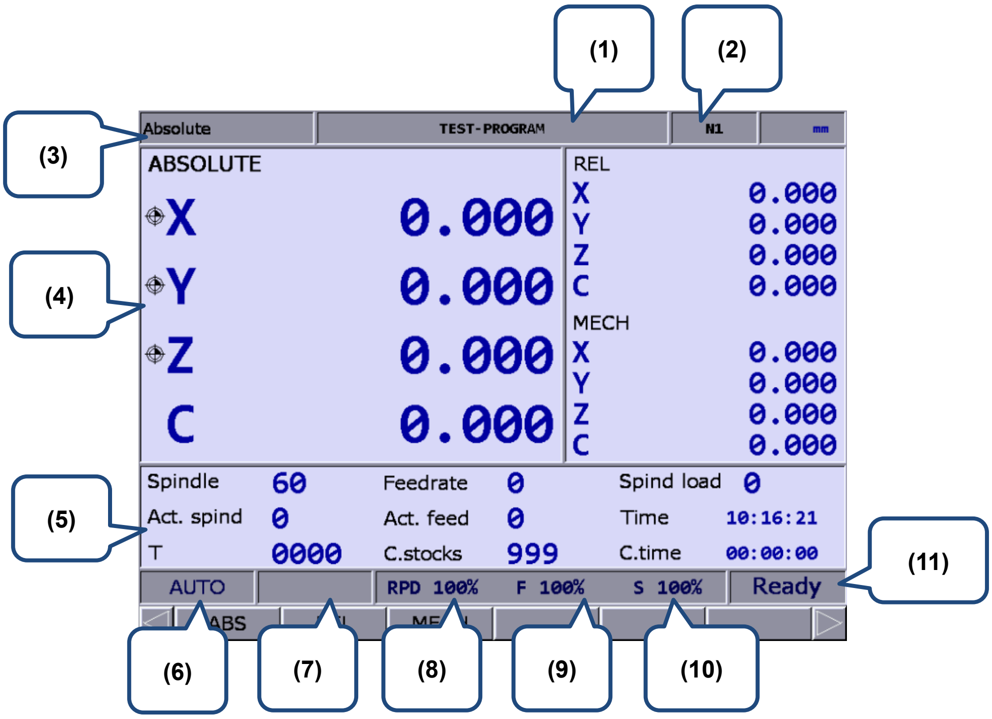
Position (POS) group 6

The POS group displays the axes positions, which are represented in absolute, relative, and machine coordinates.

6

POS group displays the axes positions, which are represented in absolute, relative, and machine coordinates. It can display the coordinates of up to three linear axes and one rotation axis according to the axis setting.

Note: bold function names in a box (such as **POS**) mean the keys on machine operation panel A; bold function names (such as **CLR ALL**) mean the function keys of F1 - F6.



(1) Name of current program

(2) Currently executed program line

(3) Current group function

(4) Current coordinates

(5) Machining information:

Spindle speed: command value

Cutting feedrate: command value

Spindle load rate: %

Actual speed: rpm

Actual feedrate: mm/min

System time

Tool number (T)

Count of machining operations

Cutting time

(6) Current system mode

(7) Alarm display

(8) Rapid traverse override

(9) Feedrate override

(10) Spindle override

(11) System status

6

6.1 Absolute coordinates

Absolute coordinates refer to the program origin of G-code, which you can use to check whether the movement specified in a program block is identical to the actual movement. The operation steps are as follows.

(1) Press **POS** to display the POS group screen and the available functions include absolute coordinates (ABS), relative coordinates (REL), and machine coordinates (MECH).
(2) Press **ABS** to enter the absolute coordinate screen.

6.2 Relative coordinates

Relative coordinates indicate the moving distance from the origin. The operation steps are as follows.

(1) Press **POS** to display the POS group screen and the available functions include absolute coordinates (ABS), relative coordinates (REL), and machine coordinates (MECH).
(2) Press **REL** to enter the relative coordinate screen.
(3) The functions available in the 2nd layer function bar include:

CLR ALL: clear the relative coordinate values of all axes.

CLR X: clear the relative coordinate value of X axis.

CLR Y: clear the relative coordinate value of Y axis.

CLR Z: clear the relative coordinate value of Z axis.

CLR A: clear the relative coordinate value of A axis.

CLR B: clear the relative coordinate value of B axis.

CLR C (next page): clear the relative coordinate value of C axis.

Note: the clear functions for the axes (X, Y, Z, A, B, and C) are available only when you connect the axes.

6.3 Machine coordinates

Machine coordinates are defined according to the mechanism. The coordinate data is neither removable nor changeable due to the selected workpiece coordinate system. The operation steps are as follows.

(1) Press **POS** to display the POS group screen and the available functions include absolute coordinates (ABS), relative coordinates (REL), and machine coordinates (MECH).
(2) Press **MECH** to enter the machine coordinate screen.

(This page is intentionally left blank.)

Program (PRG) group

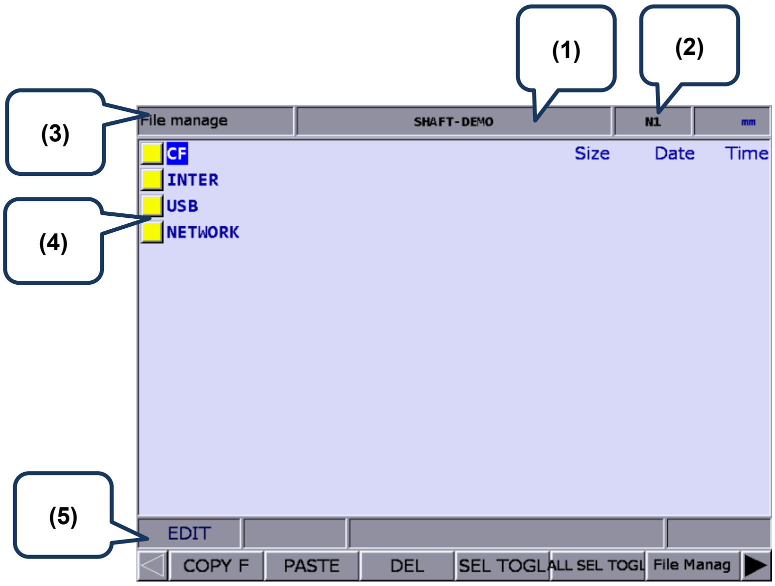
7

The PRG group provides functions of file management and program editing for G-code and macro files. In addition, some functions are specific to particular system modes.

You can manage and edit G-code and macro files with PRG group functions. **File manage** includes three layers: (1) CF (CF card), INTER (internal memory), USB (USB disk), and NETWORK; (2) folders and G-code files; (3) G-code files.

Some specific functions are available in particular system modes. For example, you can use the function of break line search in AUTO mode or you can enter and execute a program in MDI mode.

Note: bold function names in a box (such as **POS**) mean the keys on machine operation panel A; bold function names (such as **CLR ALL**) mean the function keys of F1 - F6.



(1) Name of current program
(2) Currently executed program line
(3) Current group function
(4) Disk options
(5) Current system mode

Set the system to EDIT mode and press **PRG** on machine operation panel A to display the PRG screen. In the File manage screen, you can press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor, press **ENTER** to enter the second or third layer, and then select a G-code file.

After selecting the G-code file, press **ENTER** to open the file and enter the edit screen. Press ↑ and ↓ (scroll the screen up or down by 1 line), and **PAGE UP** and **PAGE DN** (scroll the screen up or down by 20 lines) to display the file content.

Note: the suggested specifications for the USB disk is as follows.

USB disk specification	
Disk format	FAT32
Disk capacity	As required

7.1 Ethernet setting

You can use Ethernet to connect to the PC to enable remote communication. You can use the CNCNetwork software to manage the online files of multiple NC controllers with one PC, enabling data sharing and file management with the PC, and transmission-along-with-machining (DNC).

Set the communication protocol between the NC system and PC by referring to Section 12.7.1 before using the network function. The following gives simple instructions.

Set the protocol of the NC system by going to **PAR** > **ETH.**.

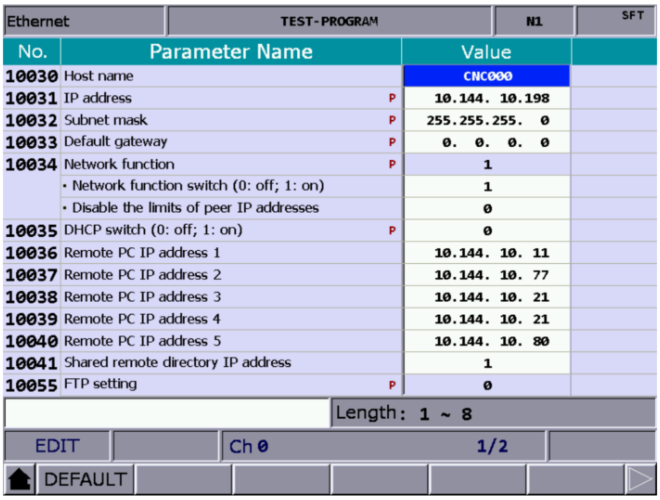


Figure 7.1.1

Network setting parameter		
Number	Name	Setting range or format
10030	Host name	Character length: 1 - 8 Actual setting: 1 - 8 characters
10031	IP address	Character length: ××× · ××× · ××× · ××× Actual setting: 192 · 168 · 0 · 2
10032	Subnet mask	Character length: ××× · ××× · ××× · ××× Actual setting: 255 · 255 · 255 · 0
10033	Default gateway	Character length: ××× · ××× · ××× · ××× Actual setting: 0 · 0 · 0 · 0
10034	Network function switch	Character length: 0 - 1 Actual setting: 1
10035	DHCP switch	Character length: 0 - 1 Actual setting: 0
10036	Remote PC IP address 1	Character length: ××× · ××× · ××× · ××× Actual setting: 192 · 168 · 0 · 1
10037	Remote PC IP address 2	Character length: ××× · ××× · ××× · ××× Actual setting: 0 · 0 · 0 · 0
10038	Remote PC IP address 3	Character length: ××× · ××× · ××× · ××× Actual setting: 0 · 0 · 0 · 0

Network setting parameter		
Number	Name	Setting range or format
10039	Remote PC IP address 4	Character length: ××× · ××× · ××× · ××× Actual setting: 0 · 0 · 0 · 0
10040	Remote PC IP address 5	Character length: ××× · ××× · ××× · ××× Actual setting: 0 · 0 · 0 · 0
10041	Shared remote directory IP address	Character length: 0 - 5 Actual setting: 0

Set the protocol of PC by setting Internet Protocol (TCP/IP) Properties on the PC operating system (as shown in Figure 7.1.2) or going to **CNCNetwork** > **Options**.

Network setting on PC:

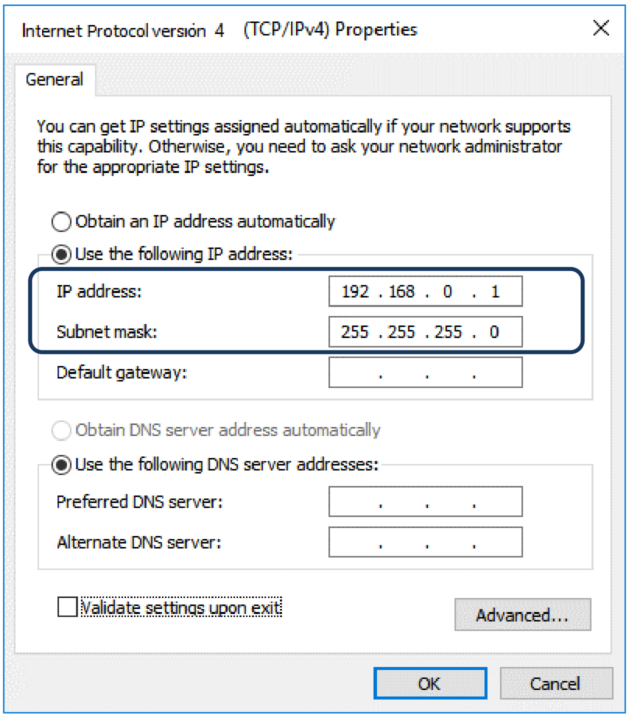


Figure 7.1.2

Steps:

(1) Select the check box for **Use the following IP address** and enter the following in sequence:

IP address: **192 . 168 . 0 . 1**

Subnet mask: **255 . 255 . 255 . 0**

(2) Click **OK** to finish the setting.

Network settings with CNCNetwork:

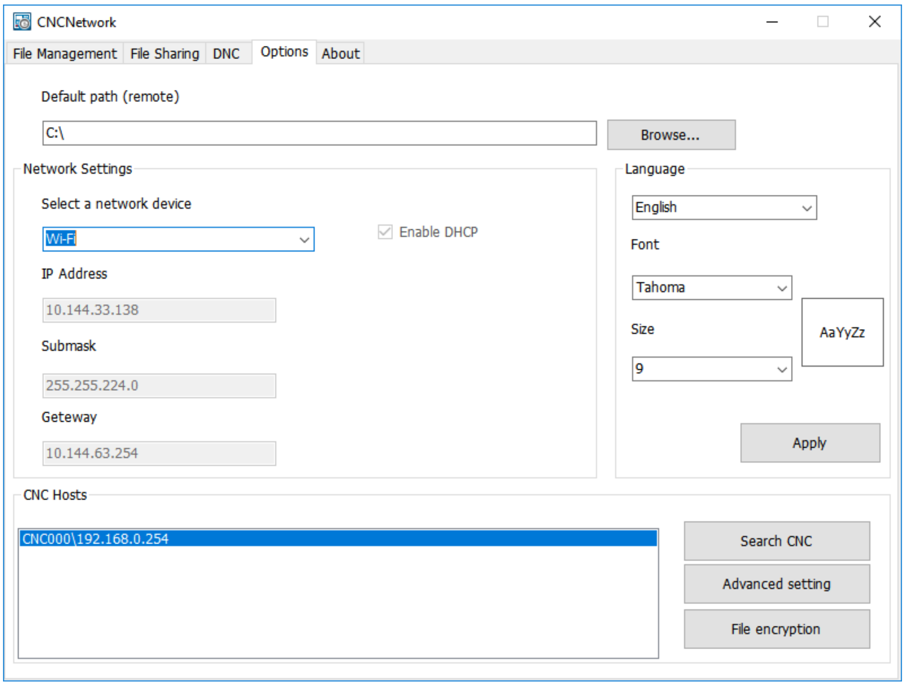


Figure 7.1.3

Steps:

(1) Execute CNCNetwork software and go to **Options** screen.

(2) Click **Search CNC** to connect to the CNC with the above settings.

7

DNC operation:

Execute CNCNetwork, open the file to be shared in the File Sharing screen, and then you can execute the G-code file while it is being processed (DNC operation) using Ethernet.
No additional disk space is required for file storage as only the path of the shared files is recorded. The connection steps are as follows.

(1) Use Ethernet communication to set the Internet connection between PC and NC.
(2) Execute CNCNetwork.
(3) Click the **DNC** tab.

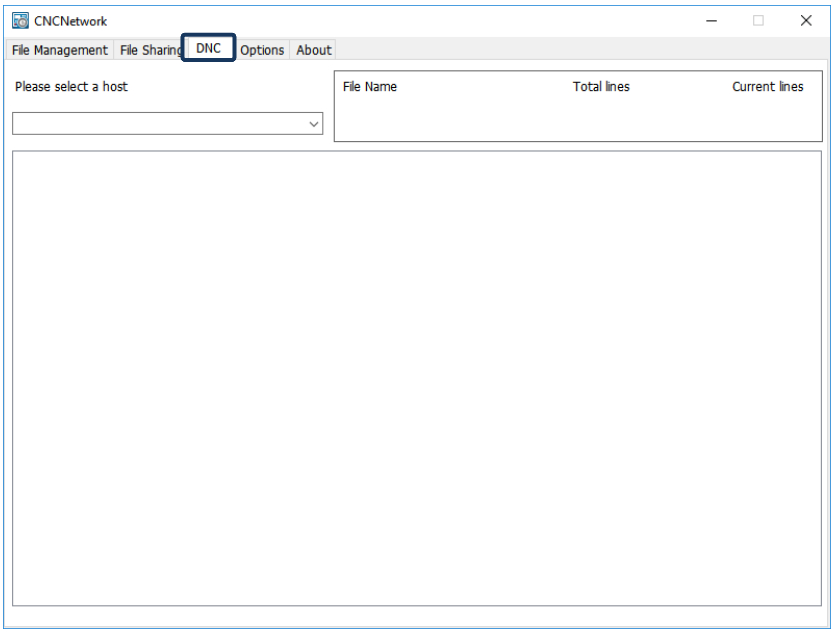


Figure 7.1.4

(4) Set the system to EDIT mode and go to **File manage** > **NETWORK**.

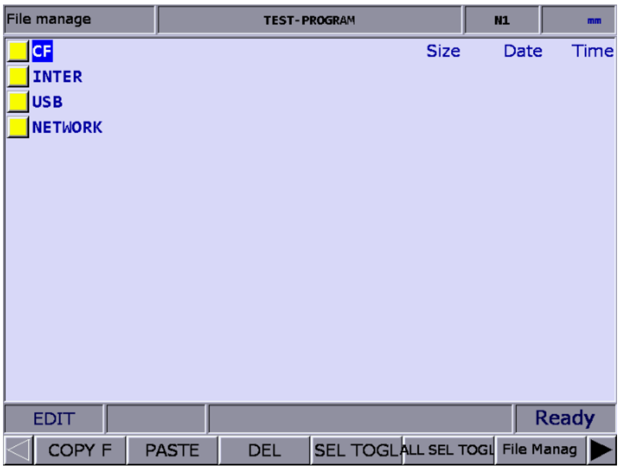


Figure 7.1.5

(5) The screen displays the shared files. Select and open the G-code file that has been set to be shared.
(6) Set the system to AUTO mode, press **CYCLE START**, and the system executes the G-code file by DNC operation. The execution method is the same as that for general files.

(7) During DNC operation, file information is displayed in the **DNC** screen of CNCNetwork. The information includes name of the connecting system, name of the file that executes DNC, total number of lines, number of line being executed, and file content (the content is scrolled down along with the execution progress, as shown in Figure 7.1.6).

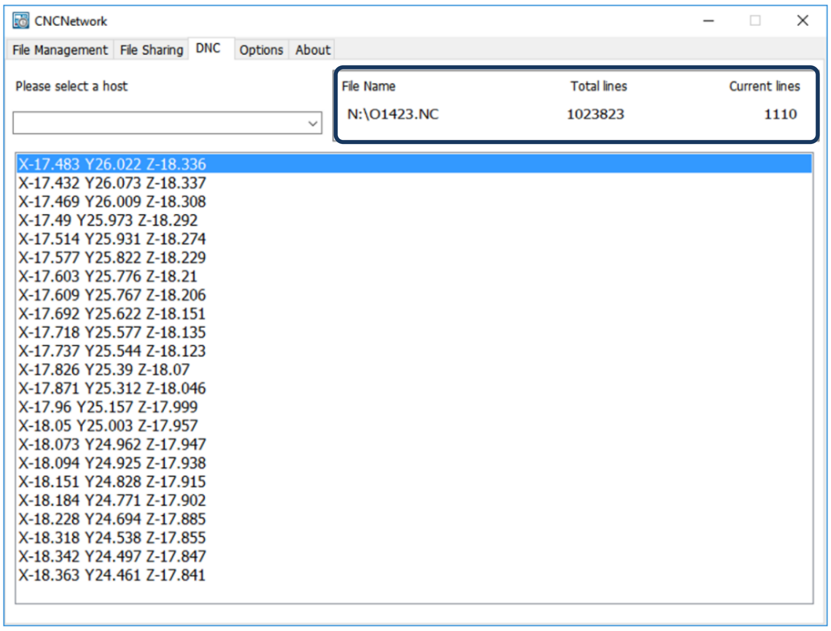


Figure 7.1.6

7.2 Create a new file (NEW F)

In EDIT mode, you can use this function to create a new G-code file from the controller interface. The operation steps are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) In the File manage screen, press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor to the destination of the disk for file creation (for example, the 2nd or 3rd layer of CF or USB directory).

(4) Press ▶ to display the function bar on the next page.

(5) Press **NEW F** and a dialog box appears for you to enter the file name.

(6) Enter alphanumeric characters (no symbols) in the dialog box, press **ENTER**, and a new file is created.

File format specifications	
Format of machining file name (G-code)	No restrictions on the format of main program names (names of each file should be unique in the same directory)
	O + 0001 to 8999 (subprograms)
Format of macro file name (O macro)	O + 9000 to 9999
Remarks in file name	Suffix '-' and alphanumeric characters in sequence to the file name
Valid format of filename extension	.NC .ANC .CNC .PIM .TAP .PTP .UOO .DEMO
Format of M macro file name	M + 10000 to 29999
Format of G macro file name	G + 30000 to 49999
Maximum allowable character length of file name	31
Storage location	2nd or 3rd management layer
Invalid symbols in file name	* / \ ｜ < > ? " :

Note:

1. Names of each file should be unique in the same directory. For example, O0001 and O1 are regarded as the same.
2. The File manage screen only displays general machining files. Macro files can be displayed by setting the parameter Pr50 Macro file display.
3. Multiple dots can be used in the file name of a G-code file whereas the last one should come with a valid format of file extension, such as "1.1.1.1.**NC**".

7.3 Copy files (COPY F)

You can use this function to copy the existing files from all disk drives. The operation steps are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) In the File manage screen, **press** ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor to the destination of the disk for file copying (for example, the 2nd or 3rd layer of CF or USB directory).

(4) Move the cursor to the file to be copied.

(5) Press **COPY F** and then **PASTE** to validate the execution.

7.4 Paste files (PASTE)

As described in Section 7.3, you should use this function together with the copy function to complete file copying. It is one of the functions of File manage in PRG group. The operation steps follow the descriptions in Section 7.3.

(1) Press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor to the location of the disk, directory, or layer for pasting the file.

(2) Enter the directory, press **PASTE**, and a dialog box appears for you to enter the file name. Enter a new file name or follow the original one, and press **ENTER** to complete the execution of file copying and pasting.

Note:

1. The specification of file naming for this function is the same as that of the file creation function. That is, file names of each file should be unique.
2. If you do not execute **COPY F** before using the **PASTE** function, an error dialog box appears to remind you to copy a file first, and thus the paste execution is invalid.
3. Follow the same operation steps to copy the files from the USB disk to the CF card.

7.5 Delete (delete files and directories)

You can use this function to delete the G-code files and directories in the second layer of **File manage**.

7

The operation steps are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) In the File Manage screen, press ⇧ and ⇩ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the second or third layer of the disk.

(4) Move the cursor to the directory or file to be deleted.

(5) Press **DEL** and a dialog box appears for you to confirm the execution. Enter “Y” and press **ENTER** to delete.

Note: the file or directory cannot be recovered once being deleted.

7.6 Select / cancel selection of multiple files (SEL TOGL / CANCEL & ALL SEL TOGL / CANCEL)

In addition to copying or deleting a single file, you can use SEL TOGL / CANCEL to select or cancel the selection of multiple files for copying, pasting, or deleting the files. The operation steps for copying and pasting multiple files are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) Enter the directory where you desire to select the files.

(4) In the File manage screen, press ⇧ and ⇩ or **PAGE UP** and **PAGE DN** to move the cursor to the file to be selected. To select a file, press **SEL TOGL** (as shown in Figure 7.6.1). To cancel the selection, press **CANCEL**. To select all files, press **ALL SEL TOGL**. To cancel the selection of all files, press **CANCEL**.

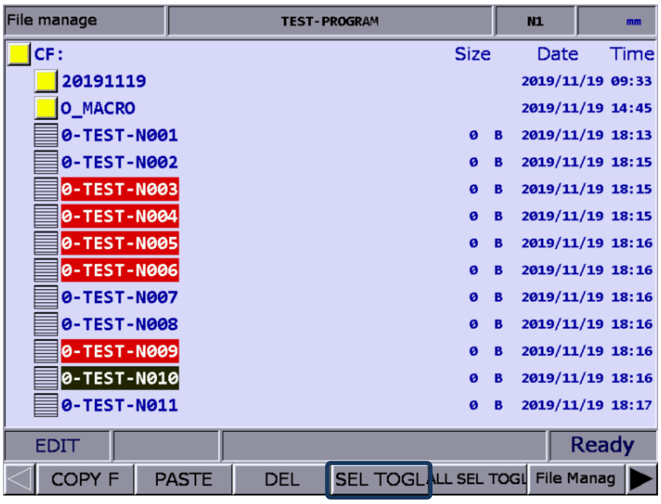


Figure 7.6.1

(5) After completing the selection, press **COPY F**.

(6) Go to another directory and press **PASTE** to paste the selected files, as shown in Figure 7.6.2.

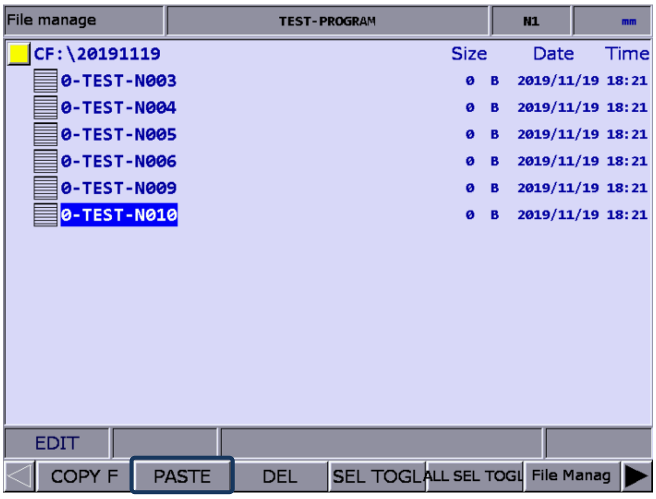


Figure 7.6.2

The operation steps for deleting multiple files are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) Enter the directory where you desire to select the files.

(4) In the File manage screen, press ⬆ and ⬇ or **PAGE UP** and **PAGE DN** to move the cursor to the file to be selected. To select a file, press **SEL TOGL**. To cancel the selection, press **CANCEL**.

(5) After selecting multiple files, press **DEL,** and a dialog box appears for you to confirm the execution (as shown in Figure 7.6.3). Enter “Y” and press **ENTER** to delete.

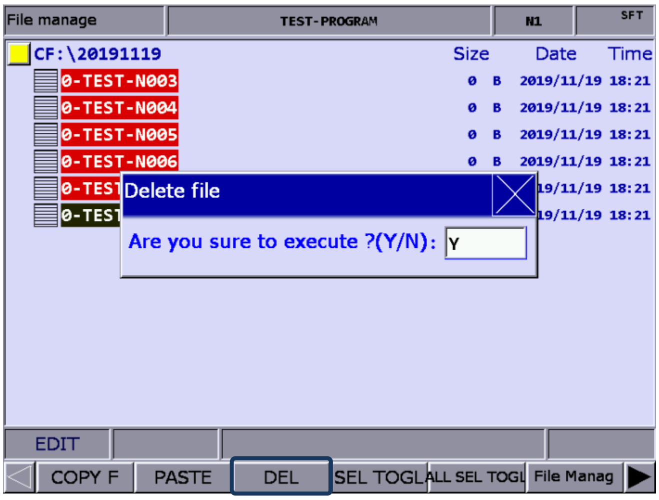


Figure 7.6.3

Note:

1. After copying multiple files, you should paste them to another directory. If you paste the files in the same directory, a dialog box appears to remind you to select another directory, and the execution is invalid.
2. When the names of the files to be pasted are identical to those of the original files, a dialog box appears for you to decide whether to overwrite the original files. Enter “Y” to replace the original files with the new ones; enter “N” or press **EXIT** to keep the original files.

7.7 Rename files (RENAME)

You can use this function to rename the file after creating the file. The operation steps are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) In the File manage screen, press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor to the destination of the disk (for example, the 2nd or 3rd layer of CF or USB directory).

(4) Press ▶ to display the function bar on the next page.

(5) Move the cursor to the file to be renamed, press **RENAME**, and a dialog box appears for you to enter the file name.

(6) Enter a name that is not identical to the file names in the directory, and press **ENTER** to complete renaming the file.

Note:

1. You can only create G-code files in the 2nd and 3rd layers of File manage but not in the 1st layer.
2. The format specification of file name for file renaming is the same as that of file creation. If you enter a name that is already used for another file in the directory when renaming, an error dialog box appears, and the execution is invalid.

7.8 Create directories (FOLDER)

This function is for creating a directory for G-code files in the 2nd layer of File manage, which is only available in the 2nd layer of File manage. Accordingly, the 2nd layer of File manage can contain both directories and G-code files. The operation steps are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) Press ▶ to display the function bar on the next page.

(4) In the 2nd layer of File manage, press **FOLDER**, and a dialog box appears for you to enter the directory name.

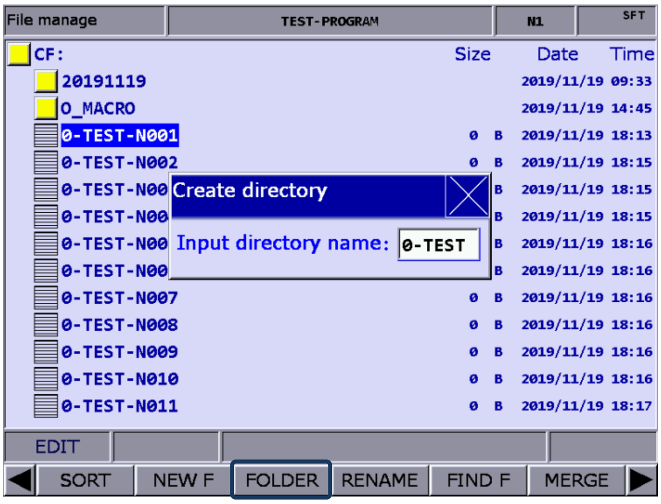


Figure 7.8.1

(5) After entering the directory name, press **ENTER** to complete creating the directory.

By following the steps above, you can create a new directory in the 2nd layer of File manage, and create and edit G-code files in this directory (the 3rd layer).

Directory format specifications	
Format of directory name	Not limited to alphanumeric characters
Maximum number of characters of directory name	31
Storage location	2nd management layer

7.9 Search for files (FIND F)

You can use this function to quickly search for the target file by file name and open it when there are a number of G-code files. The operation steps are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) In the File manage screen, press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the 2nd or 3rd layer of the disk.

(4) Press ▶ to display the function bar on the next page.

(5) Press **FIND F** and a dialog box appears for you to enter the file name to be searched. After entering the file name, press **ENTER** to search for and open the file.

Note:

1. You can only search for files in the same directory with this function.
2. Enter the complete file name to accurately search for and open the file.

7.10 File merging (MERGE)

You can use this function with the file copying function to merge the program content of two different G-code files. The operation steps are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) In the File manage screen, press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the 2nd or 3rd layer of the disk.

(4) Select the G-code file to be copied.

(5) Press **COPY F** and the content is saved to the system’s clipboard.

(6) Move the cursor to the directory which contains files to be merged.

(7) Press ▶ to display the function bar on the next page.

(8) Press **MERGE** and a dialog box appears for you to enter the name of the merged file. After entering the name, press **ENTER** to open the file.

(9) Move the cursor to the line where you desire to paste the program content, and press **PASTE**.

(10) To save the merged file, switch to different system modes, open another file, or press **RESET**.

7.11 Sequencing (SORT)

7

This function is for sequencing the directories or files in a directory by a specified order, facilitating the operation of file search or management.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) In the File manage screen, press ⬆ and ⬇ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the 2nd or 3rd layer of the disk.

(4) Press ▶ to display the function bar on the next page.

(5) Press **SORT** to display the function bar in the 2nd layer.

(6) Press **NAME**, and the directories and files are displayed by the sequence of number > English letters (from top to bottom). Press **NAME** again, and they are displayed by the sequence of English letters > number (from top to bottom).

(7) Press **SIZE** to display the directories and files by the file size from small to large (from top to bottom). Press **SIZE** again to display them by the file size from large to small (from top to bottom).

(8) Press **DATE** to display the directories and files by the date from most recent to earlier (from top to bottom). Press **DATE** again to display them by the date from earlier to most recent (from top to bottom).

7.12 Convert DXF files

You can find this function from the function bar in the File manage screen. After selecting the DXF file, enter relevant parameter settings to convert them into an executable G-code file.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) Press ▶ twice to display the function bar on the third page.

(4) Press **DXF** to display the DXF file manage screen.

(5) In the File manage screen, press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to select the DXF file to be converted.

(6) After selecting the DXF file, you are redirected to the screen for setting the relevant parameters, as shown in the following figure.

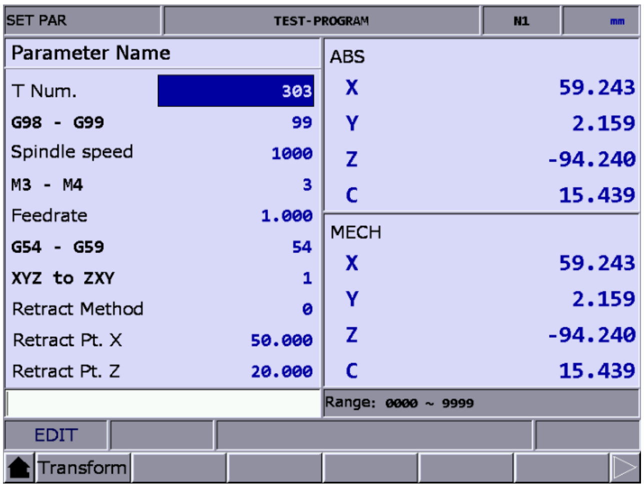


Figure 7.12.1

(7) After finishing setting the parameters, press **Transform**, and a dialog box appears for you to enter the new file name.

(8) After entering the file name, press **ENTER** to convert the DXF file into a G-code file, and the G-code file is stored in the CF directory.

(9) Then, you can execute the G-code file in AUTO mode.

7.13 Macro files

In response to the application requirements, this function is for managing the equipment-specific macro files. Upon accessing the security authorization, you can use all the editing functions described in Section 7.14. Otherwise, you can only browse the existing macro files rather than open and edit them. Contact the local distributor for authorization settings.

7.14 File editing (File edit)

You can use this function to modify or delete the content of the G-code files. After you open the file in the File manage screen, the system switches to the File edit screen. Move the cursor to any position in the program and use the alphabetic, numeric, and editing keys on machine operation panel A to edit the program. To save the file after editing the program, switch to different system modes, press **RESET**, or open another file. The operation steps for entering the File edit screen are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) Press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the 2nd or 3rd layer of the disk.

(4) Select the G-code file to be edited, and press **ENTER** to open the file and enter the edit screen.

(5) Press ↑, ↓, ←, and → to move the cursor to any position in the program.

(6) Edit the content by pressing the alphabetic, numeric, and editing keys on machine operation panel A.

(7) To save the file after finishing editing, switch to different system modes, open another file, or press **RESET**.

Specifications for editing	
Maximum number of characters of a single line	255
Supported mode	EDIT mode
Allowable file size	Below 3 MB

Note:

1. When using the File manage or File edit function, you have to set the system to EDIT mode to display the corresponding function bar. Otherwise, the PRG screen is only for viewing the currently opened program file and displays the coordinate information.
2. You can insert "( )" (parentheses) at the end of each program block in the G-code file for making notes. Do not insert parentheses in the beginning of the program block, or the block may be taken as a note and be skipped.

7.14.1 Line search (LABEL)

This function is for searching the specific line of program in the G-code file. The operation steps are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) Press ⇧ and ⇩ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the 2nd or 3rd layer of the disk.

(4) Select the G-code file to be edited, and press **ENTER** to open the file and enter the edit screen.

(5) Press ▶ to display the function bar on the next page.

(6) Press **LABEL** and a dialog box appears for you to enter the line number (by pressing the numeric keys 0 - 9).

(7) After entering the line number, press **ENTER**, and the cursor jumps to the specified line, completing the action.

Requirements for line search	
Maximum number of characters of searching string	62
Format of searching string	Specific line of the program

7.14.2 String search (STRING)

The line search function is only for searching the specific line while you can use this function to search for specific strings. The accuracy of the searching result depends on how precise the input string is. The string search function contains the function of string replacing. You can determine whether to replace a string when searching for a string, which enables you to directly replace the string on the panel screen. The operation steps are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) Press ⇧ and ⇩ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the 2nd or 3rd layer of the disk.

(4) Select the G-code file to be edited, press **ENTER** to open the file and enter the edit screen.

(5) Press ▶ to display the function bar on the next page.

(6) Press **STRING** and a dialog box appears for you to enter the string to be searched, as shown in the following figure.

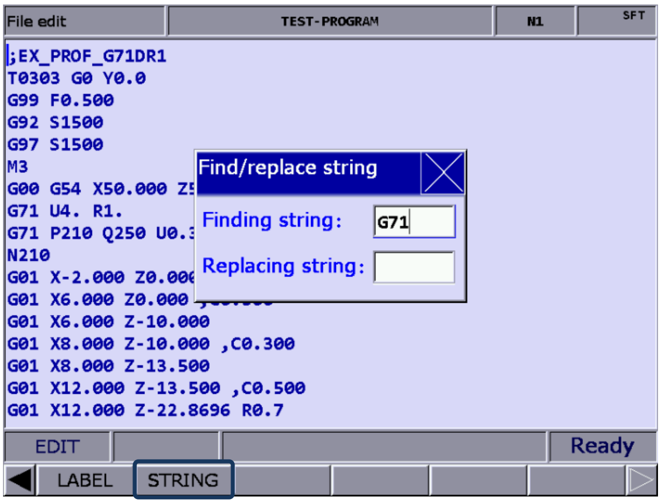


Figure 7.14.2.1

(7) After entering the string to be searched and the string to be replaced, press **ENTER** to search the string.

(8) After that, the searched string is highlighted. At the same time, “NEXT”, “PREV”, “REPLACE”, and “Replace all” are displayed on the function bar.

(9) Press **NEXT** to search the next match or press **PREV** to search the previous match.

(10) Press **REPLACE** when you desire to replace one single string. You can press **Replace all** to batch replace the matches with the new string.

(11) Press ◀ to exit the string search function and go back to the function bar of File edit.

(12) After replacing the string, ensure to save the results (by switching to different system modes, opening another file, or pressing **RESET**).

Requirements for string replacing	
Supported mode	EDIT mode
Allowable file size for editing and replacing	Below 3 MB

7

7.14.3 Edit a section of program (B START / B END)

To edit a section of a program, you can use the B START / B END functions to specify the start and end of the content to be edited. Then, you can delete, copy, and paste the selected program content as required, which simplified the editing process. The operation steps are as follows.

(1) Set the system to EDIT mode.

(2) Press **PRG** to switch to the PRG screen.

(3) Press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the 2nd or 3rd layer of the disk.

(4) Select the G-code file to be edited, press **ENTER** to open the file and enter the edit screen.

(5) Press ↑, ↓, ←, and → to move the cursor to the start of the section to be edited and press **B START**.

(6) Move the cursor to the end of the section to be edited and press **B END**. See the following figure for the screen for the selected section.

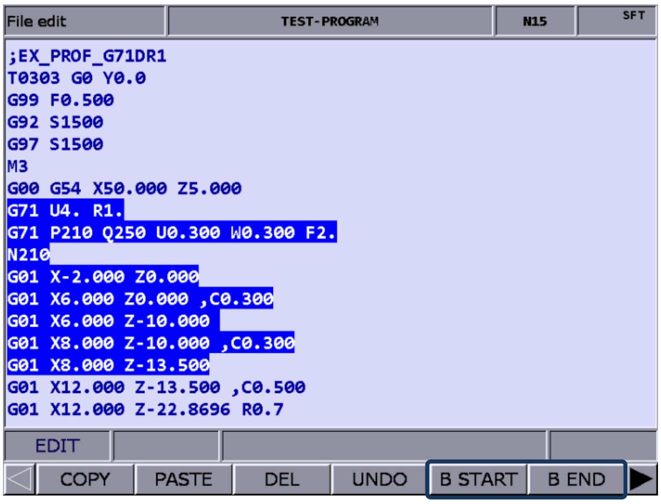


Figure 7.14.3.1

(7) Follow Steps (5) - (6) then press **DEL**, and you can delete the selected content.

(8) Follow Steps (5) - (6) then press **COPY** to copy the selected content. Move the cursor to where you wish to paste the copied content and press **PASTE** to insert the content.

7.14.4 Delete (lines and blocks)

You can use this function to delete a single line of program where the cursor is located. And you can use this function with B START and B END to delete a whole section of the program. The operation steps are as follows.

(1) Set the system to EDIT mode.
(2) Press **PRG** to switch to the PRG screen.
(3) Press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the 2nd or 3rd layer of the disk.
(4) Select the G-code file to be edited, and press **ENTER** to open the file and enter the edit screen.
(5) Move the cursor to the line to be deleted and press **DEL** to delete the line of program.
(6) Follow Step (7) in Section 7.14.3 to delete a whole section of a program.

7.14.5 Copy (lines and blocks) and paste

Move the cursor to the specified line and press **COPY**, which takes effect when used with the PASTE function. You can use this function to copy a single line or a section of the program content. The operation steps are as follows.

(1) Set the system to EDIT mode.
(2) Press **PRG** to switch to the PRG screen.
(3) Press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the 2nd or 3rd layer of the disk.
(4) Select the G-code file to be edited, and press **ENTER** to open the file and enter the edit screen.
(5) Move the cursor to the line of program to be copied and press **COPY**.
(6) Move the cursor to the position to paste the copied content, and press **PASTE** to paste the line of program to that position.
(7) Follow Step (8) in Section 7.14.3 to copy a whole section of a program.

7.14.6 Undo

During program editing, use this function to undo the previous edit. You can repeatedly use this function for undoing up to 7 previous steps. The operation steps are as follows.

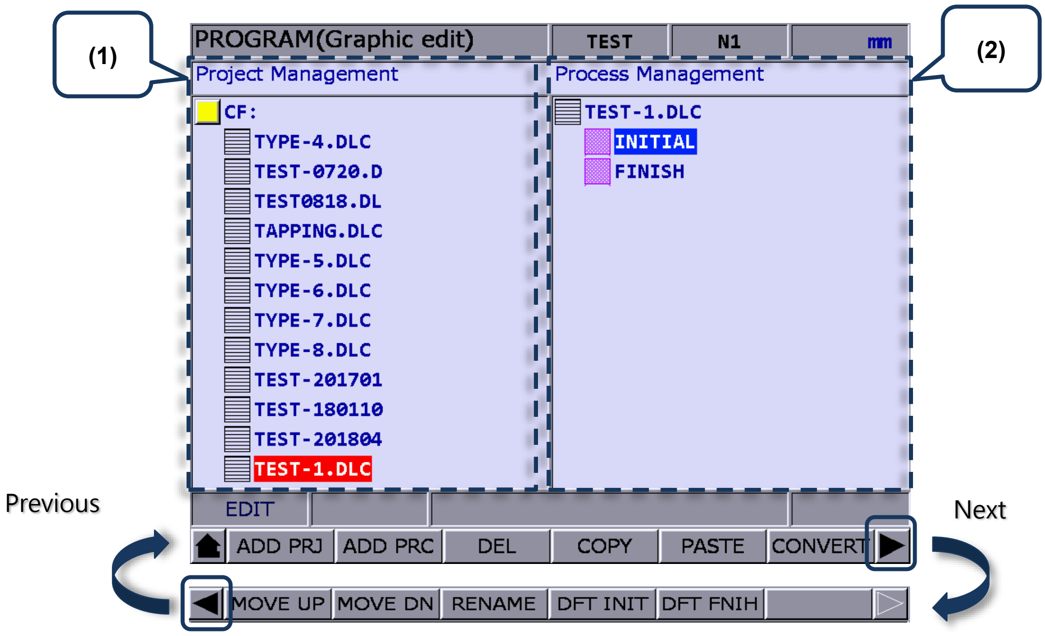
(1) Set the system to EDIT mode.
(2) Press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor, and press **ENTER** to enter the 2nd or 3rd layer of the disk.
(3) Select the G-code file to be edited, and press **ENTER** to open the file and enter the edit screen.
(4) After editing the program, press **UNDO** to undo the previous edit.

7.15 Graphic edit (lathe system)

7.15.1 Objective

This function provides an interface with graphics for you to select the machining procedure and enter relevant machining parameters to generate a tool path program, which replaces manual programming and calculation as well as planning the machining path with CAM.

7.15.2 Project management



(1) Project Management screen; (2) Process Management screen

■ **Add a new project (ADD PRJ)**

Enter the Graphic edit screen, press **ADD PRJ**, and a dialog box appears. Enter the file name and press **ENTER** to add a new project file with the filename extension as .DLC. You can repeatedly use the project file. Simply open the .DLC project file in the Project Management screen, and you can get all the previously set procedures, and modify or export the machining program.

If you press **ADD PRJ** in the Process Management screen, a dialog box appears to remind you to return to the Project Management screen. Press ← to continue the operation.

- **Add a new procedure (ADD PRC)**

Open a project file before you add a new procedure. If no project file is selected, a dialog box displaying “Please open a file!” appears when you press **ADD PRC**.

Select a project file and press **ENTER** to enter its corresponding Process Management screen. The system automatically generates two default machining procedures for each created project file, INITIAL and FINISH, which start and end the program respectively. You can add or modify procedures between them.

If you desire to select another project file, press ⬅ to exit the Process Management screen.

- **Delete (DEL)**

Press this key to delete the selected project file or procedure, except INITIAL and FINISH. Move the cursor to the project file or procedure to be deleted, press **DEL**, and a dialog box appears. Enter “Y” to confirm the deletion.

- **Copy and paste (COPY & PASTE)**

These two functions are for copying and pasting the project file or procedure. Move the cursor to the file to be copied, press **COPY** and then **PASTE**, and a dialog box appears. Enter the new file name and press **ENTER** to complete the action.

- **Convert a file (CONVERT)**

Press this key to convert the selected .DLC project file into an .NC file. After converting, the system automatically returns to the Program Management screen.

Important:

(1) If you press **CONVERT** without selecting the disk type in the File manage screen, a dialog box appears to remind you to return to the File manage screen. Press to return to the screen and select the disk for storing the file before you start the conversion.

(2) Select a .DLC file and press **ENTER** to open that file. If no project file is selected, a dialog box displaying “Please open one DLC file!” appears.

- **Sequencing (MOVE UP & MOVE DN)**

The two functions are only available in the Process Management screen. Select a project file, enter its corresponding Process Management screen, move the cursor to the procedure which you desire to change its sequence, and then press **MOVE UP** or **MOVE DN** to change its sequence. Note that this function cannot change the sequence of INITIAL and FINISH.

- **Rename (RENAME)**

Press this key to rename the selected project file or procedure, except INITIAL and FINISH. Move the cursor to the project file or procedure to be renamed, press **RENAME**, and a dialog box appears. Enter the file name and press **ENTER** to change the file name.

- **Default (DFT INIT & DFT FNIH)**

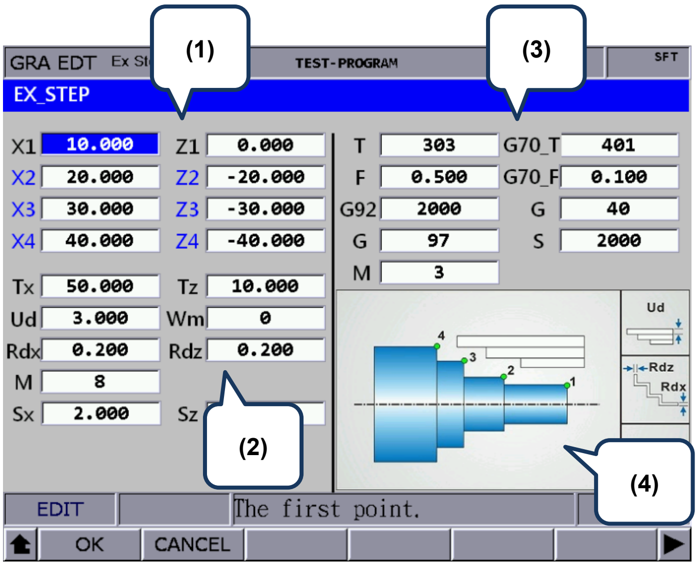
Press **DFT INIT** to change the default content of INITIAL. Press **DFT FNIH** to change the default content of FINISH. The changed content of INITIAL and FINISH will be displayed in the project that is added next time. The content of the existing projects remains unchanged.

7.15.3 Description of graphic edit procedures for lathe system

Sorting of currently available procedures

- Graphic edit for lathe
  - Step turning
    - 1. External step turning [EX_STEP]
    - 2. Internal step turning [IN_STEP]
    - 3. Face step turning [FC_STEP]
  - Taper turning
    - 1. External taper turning [EX_TAPER]
    - 2. Internal taper turning [IN_TAPER]
  - Threading
    - 1. External threading [EX_THRD]
    - 2. Internal threading [IN_THRD]
  - Grooving
    - 1. External grooving [EX_GRV]
    - 2. Internal grooving [IN_GRV]
    - 3. Face grooving [FC_GRV]
  - Arc turning
    - 1. Outer convex turning [EX_ROUT]
    - 2. Inner convex turning [IN_ROUT]
    - 3. Outer concave turning [EX_RIN]
    - 4. Inner concave turning [IN_RIN]
  - Profile turning
    - 1. External profile turning [EX_PROF]
    - 2. Internal profile turning [IN_PROF]

■ External step turning [EX_STEP]



(1) Input fields for coordinates of each step point

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
3rd point	X3_, Z3_	4th point	X4_, Z4_

These 8 input fields are for setting the three-step contour machining. If the application requires less than three steps, you can leave the fields of items in blue blank. To delete the values in the input fields, press **BACK SPACE** or **DEL** and then press **ENTER** to confirm. Refer to section (4) in the above figure for the position of each point.

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning (absolute coordinate of X axis)	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Rdx_	Allowance of finish turning (absolute coordinate of X axis)	Rdz_	Allowance of finish turning (absolute coordinate of Z axis)
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

- The input values for Ud, Rdx, and Rdz must be greater than 0.
- Wm is for setting the cutting mode. Set Wm to 0 to perform a complete turning process including rough and finish turning. Set Wm to 1 to perform rough turning and keep the allowance of finish turning. Set Wm to 2 to perform finish turning once according to the given path.
- Tx and Tz are for setting the tool change position. After finishing a machining procedure, the system moves the tool to the position specified by Tx and Tz to perform tool change for the next machining procedure.

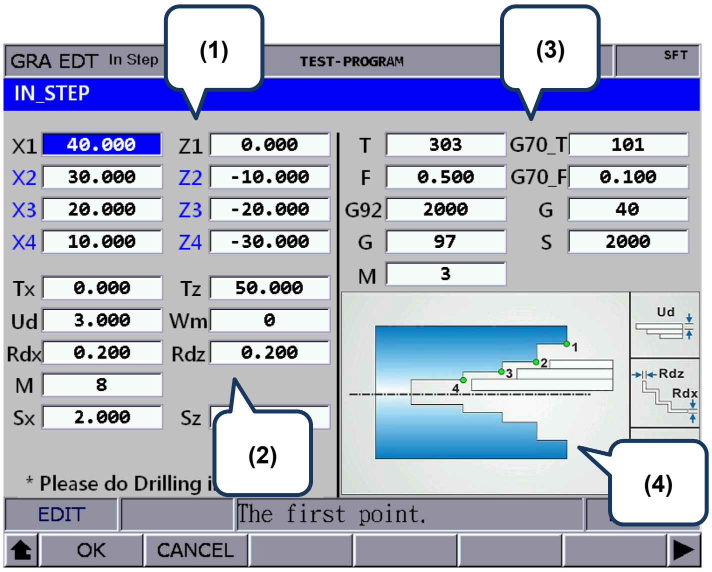
(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Item	Description	Item	Description
T_	Tool number + tool compensation number Format: TXXXX	G70_T	Tool for executing G70 Finish turning cycle
F_	Axis feedrate Unit: mm/min or mm/rev	G70_F	Feedrate for executing G70 Finish turning cycle
G92_	Maximum spindle speed	G_	Tool nose compensation (40 = disabled; 41 = left; 42 = right)
G_	Spindle speed control mode (G96 = constant; G97 = fixed)	S_	Spindle speed
M_	Spindle control (3 = forward; 4 = reverse; 5 = stop)	-	-

- T is for selecting the machining tool. Enter numbers in this field with the first two (or the first one if the number is a one-digit number) as the tool number and the last two as the compensation number. As shown in the figure above, 303 means tool No.3 is used and tool compensation No.3 is enabled.
- G70_T is for specifying the tool number for executing G70 Finish turning cycle. The format is the same as that for item T.
- G70_F is for specifying the tool feedrate for executing G70 Finish turning cycle.
- Tool nose compensation (G_): G40 / G41 / G42
  G40 means the function is disabled, G41 is tool nose radius compensation left, and G42 is tool nose radius compensation right. You have to fill the correct tool nose type and tool radius value in the tool compensation table before using this function.
- Spindle speed control mode (G_): G96 / G97
  Set this field to G96 to enable constant surface speed control and the S field is for setting the cutting speed in the unit of m/min or feed/min.
  Set this field to G97 to enable fixed spindle speed control and the S field is for setting the speed per minute in the unit of rpm.
- G92 is for setting the maximum spindle speed. If the command speed exceeds this setting, the spindle runs with this setting.

(4) Illustration

■ **Internal step turning [IN_STEP]**



(1) Input fields for coordinates of each step point (You can leave the fields of items in blue blank, which means the points do not exist.)

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
3rd point	X3_, Z3_	4th point	X4_, Z4_

Refer to section (4) in the above figure for the position of each point.

(2) Parameter settings for contour machining

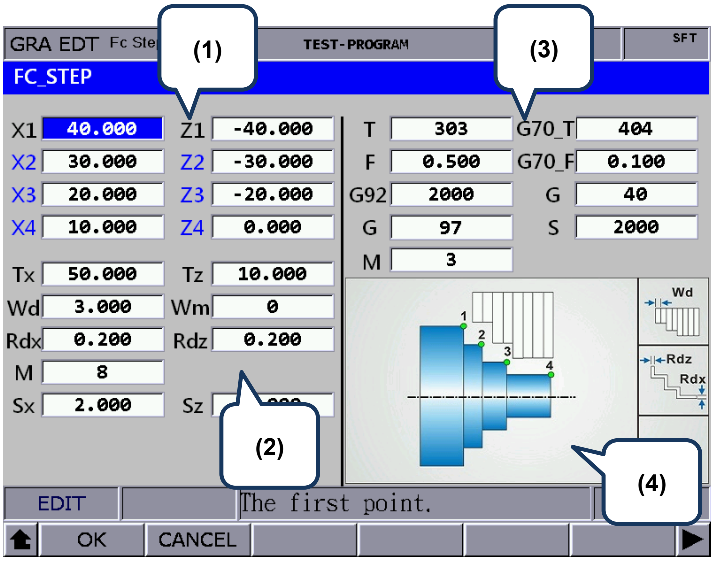
Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning (absolute coordinate of X axis)	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Rdx_	Allowance of finish turning (absolute coordinate of X axis)	Rdz_	Allowance of finish turning (absolute coordinate of Z axis)
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Item	Description	Item	Description
T_	Tool number + tool compensation number Format: TXXXX	G70_T	Tool for executing G70 Finish turning cycle
F_	Axis feedrate Unit: mm/min or mm/rev	G70_F	Feedrate for executing G70 Finish turning cycle
G92_	Maximum spindle speed	G_	Tool nose radius compensation (40 = disabled; 41 = left; 42 = right)
G_	Spindle speed control mode (G96 = constant; G97 = fixed)	S_	Spindle speed
M_	Spindle control (3 = forward; 4 = reverse; 5 = stop)	-	-

(4) Illustration

■ **Face step turning [FC_STEP]**



(1) Input fields for coordinates of each step point (You can leave the fields of items in blue blank, which means the points do not exist.)

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
3rd point	X3_, Z3_	4th point	X4_, Z4_

Refer to section (4) in the above figure for the position of each point.

(2) Parameter settings for contour machining

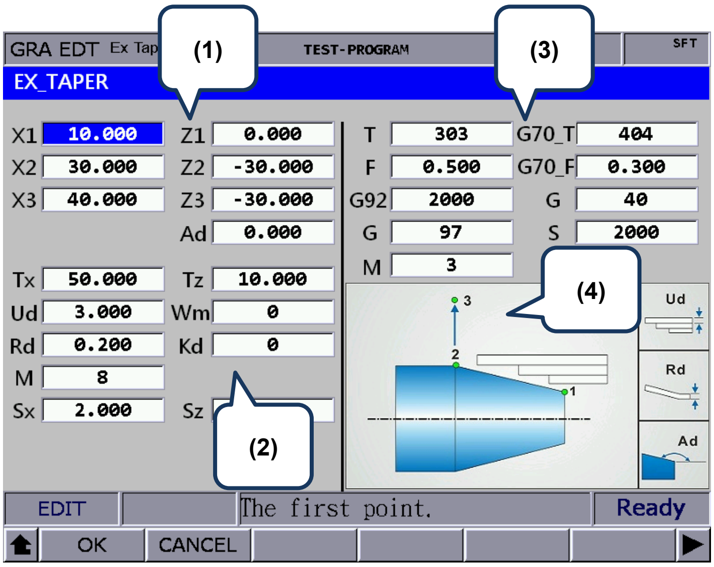
Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning (absolute coordinate of X axis)	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Rdx_	Allowance of finish turning (absolute coordinate of X axis)	Rdz_	Allowance of finish turning (absolute coordinate of Z axis)
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Item	Description	Item	Description
T_	Tool number + tool compensation number Format: TXXXX	G70_T	Tool for executing G70 Finish turning cycle
F_	Axis feedrate Unit: mm/min or mm/rev	G70_F	Feedrate for executing G70 Finish turning cycle
G92_	Maximum spindle speed	G_	Tool nose radius compensation (40 = disabled; 41 = left; 42 = right)
G_	Spindle speed control mode (G96 = constant; G97 = fixed)	S_	Spindle speed
M_	Spindle control (3 = forward; 4 = reverse; 5 = stop)	-	-

(4) Illustration

■ **External taper turning [EX_TAPER]**



(1) Input fields for coordinates of each step point (You can leave the fields of items in blue blank, which means the points do not exist.)

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
3rd point	X3_, Z3_	Ad	Taper angle

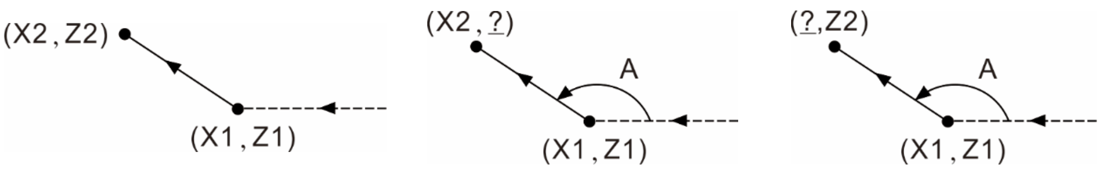
Refer to section (4) in the above figure for the position of each point.

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning (absolute coordinate of X axis)	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Rd_	Allowance of finish turning	Kd_	Taper type (0 = XZ; 1 = XA; 2 = ZA)
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

Kd: taper type

0: XZ type 1: XA type 2: ZA type

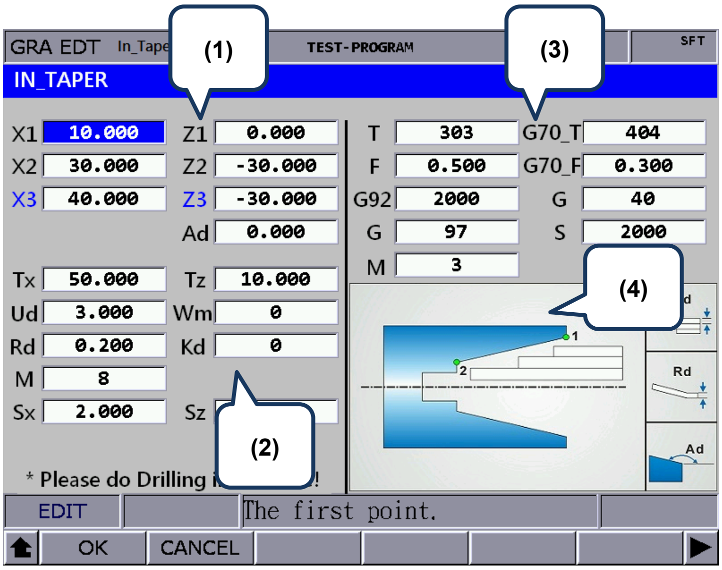


(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

■ Internal taper turning [IN_TAPER]



(1) Input fields for coordinates of each step point (You can leave the fields of items in blue blank, which means the points do not exist.)

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
3rd point	X3_, Z3_	Ad	Taper angle

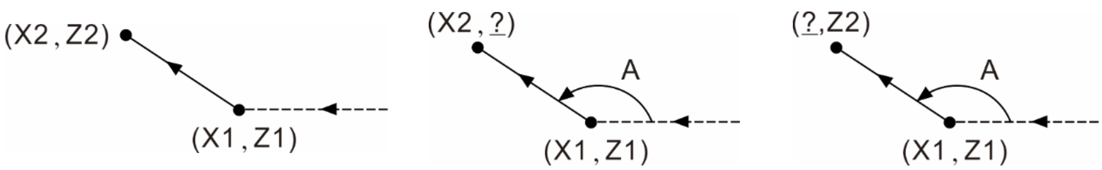
Refer to section (4) in the above figure for the position of each point.

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning (absolute coordinate of X axis)	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Rd_	Allowance of finish turning	Kd_	Taper type (0 = XZ; 1 = XA; 2 = ZA)
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

Kd: taper type

0: XZ type 1: XA type 2: ZA type

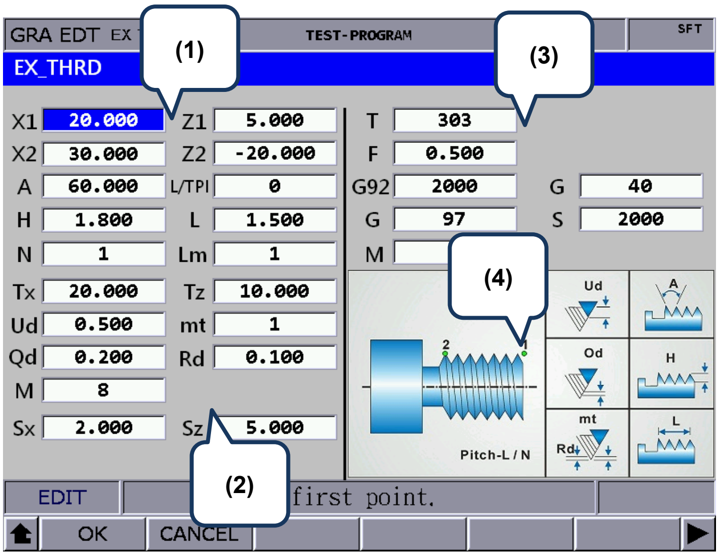


(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

■ **External threading [EX_THRD]**



(1) Input fields for coordinates of each step point

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
A	Thread angle	L/TPI	0 = thread lead 1 = number of threads per inch
H	Total thread cutting depth	L	Unit for L/TPI L = thread lead in the unit of mm/pitch TPI = number of threads per inch
N	Number of threads	Lm	Threading infeed 0: right; 1: middle; 2: left; 3: right-left shift

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Depth of the first cut	mt	Number of finish cutting
Qd_	Minimum cutting depth	Rd_	Allowance of finish turning
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

■ Ud is for specifying the depth of the first cut. Depth of the n cut (n >= 2) is automatically calculated by the system with the formula $\mathrm{Ud}\left[\sqrt{n}-\sqrt{(n-1)}\right]$.

■ Qd is for specifying the minimum cutting depth. When the cutting amount of the n cut is smaller than the amount set by Qd, the system sets the value of Qd as the feed amount.

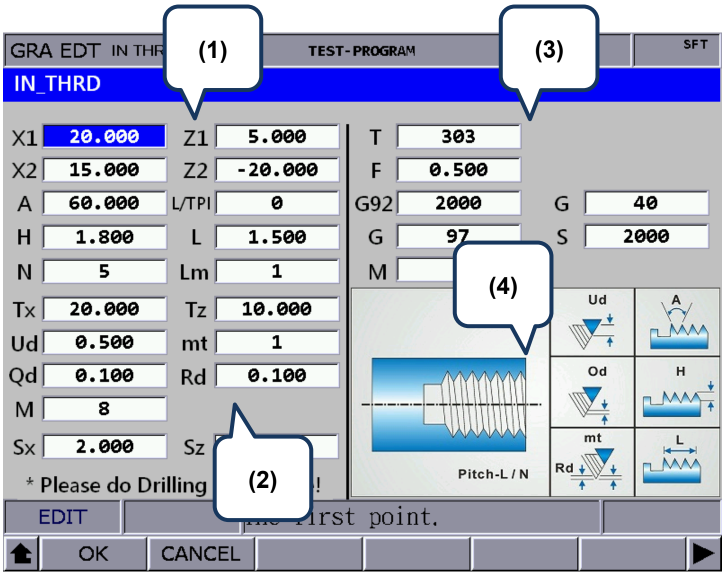
■ Rd is the allowance of finish turning and mt is the number of finish turning, so the amount of each finish turning is Rd/mt.

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

■ Internal threading [IN_THRD]



(1) Input fields for coordinates of each step point

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
A	Thread angle	L/TPI	0 = thread lead 1 = number of threads per inch
H	Total thread cutting depth	L	Unit for L/TPI L = thread lead in the unit of mm/pitch TPI = number of threads per inch
N	Number of threads	Lm	Threading infeed 0: right; 1: middle; 2: left; 3: right-left shift

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Depth of the first cut	mt	Number of finish cutting
Qd_	Minimum cutting depth	Rd_	Allowance of finish turning
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

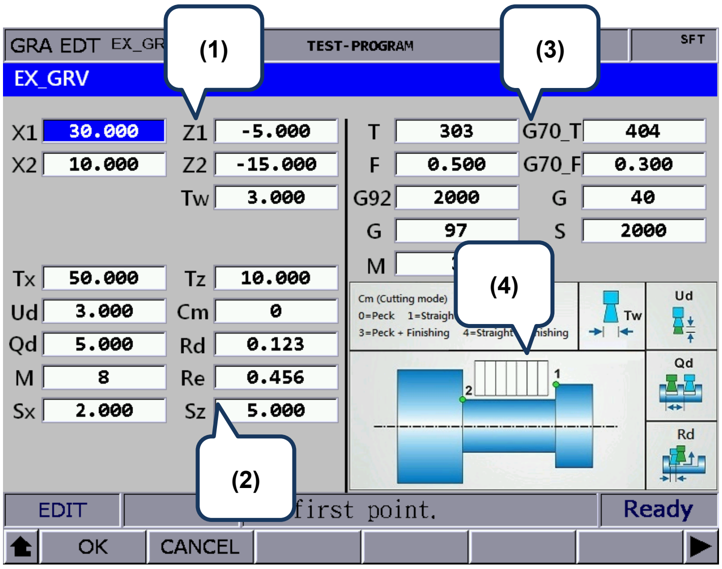
- ■ Ud is for specifying the depth of the first cut. Depth of the n cut (n >= 2) is automatically calculated by the system with the formula $\mathrm{Ud}\left[\sqrt{n}-\sqrt{(n-1)}\right]$.
- ■ Qd is for specifying the minimum cutting depth. When the cutting amount of the n cut is smaller than the amount set by Qd, the system sets the value of Qd as the feed amount.
- ■ Rd is the allowance of finish turning and mt is the number of finish turning, so the amount of each finish turning is Rd/mt.

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

■ **External grooving [EX_GRV]**



(1) Input fields for coordinates of each step point

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
Tw	Groove width	-	-

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	X axis feeding amount of each peck turning	Cm_	Cutting mode setting (0 = peck; 1 = straight; 2 = finish)
Qd_	Z axis feeding amount of each cut	Rd_	Retraction amount of Z axis after the cutting in X-axis direction is finished
M_	Coolant switch (8 = on; 9 = off)	Re_	Retraction amount of X axis after each peck turning
Sx, Sz	Safety clearance (coordinates)	-	-

■ The input values for Ud, Qd, and Rd must be greater than 0.

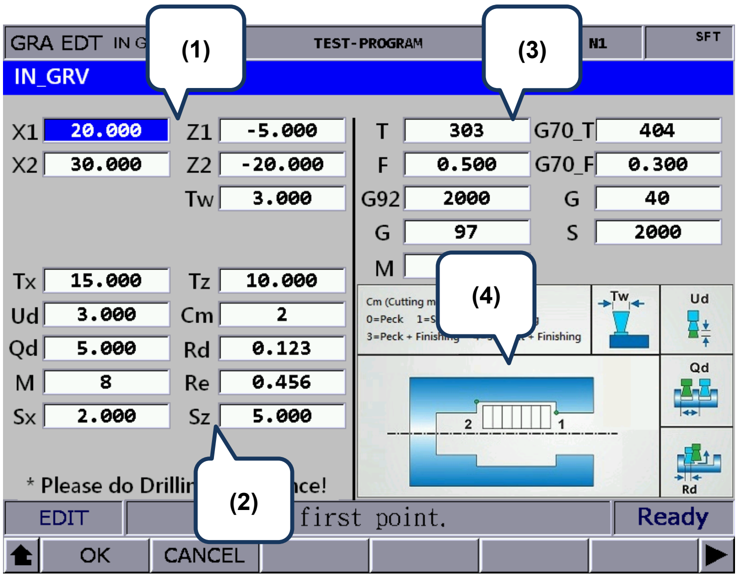
■ Cm is for setting the cutting mode. Set Cm to 0 to have the system perform peck turning. Set Cm to 1 to perform straight cutting. Set Cm to 2 to perform finish turning once according to the given path.

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

■ Internal grooving [IN_GRV]



(1) Input fields for coordinates of each step point

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
Tw	Groove width	-	-

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	X axis feeding amount of each peck turning	Cm_	Cutting mode setting (0 = peck; 1 = straight; 2 = finish)
Qd_	Z axis feeding amount of each cut	Rd_	Retraction amount of Z axis after the cutting in X-axis direction is finished
M_	Coolant switch (8 = on; 9 = off)	Re_	Retraction amount of X axis after each peck turning
Sx, Sz	Safety clearance (coordinates)	-	-

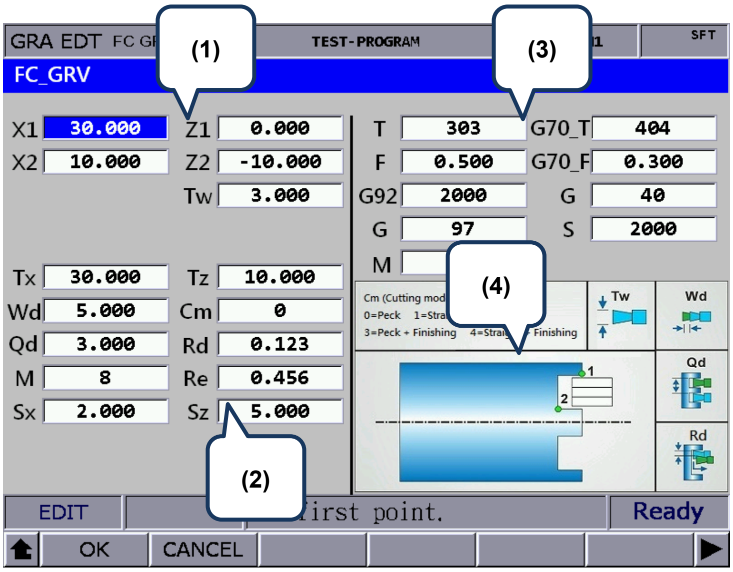
- The input values for Ud, Qd, and Rd must be greater than 0.
- Cm is for setting the cutting mode. Set Cm to 0 to have the system perform peck turning. Set Cm to 1 to perform straight cutting. Set Cm to 2 to perform finish turning once according to the given path.

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

- **Face grooving [FC_GRV]**



(1) Input fields for coordinates of each step point

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
Tw	Groove width	-	-

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Wd_	Z axis Feeding amount of each peck turning	Cm_	Cutting mode setting (0 = peck; 1 = straight; 2 = finish)
Qd_	X axis Feeding amount of each cut	Rd_	Retraction amount of X axis after the cutting in Z-axis direction is finished
M_	Coolant switch (8 = on; 9 = off)	Re_	Retraction amount of Z axis after each peck turning
Sx, Sz	Safety clearance (coordinates)	-	-

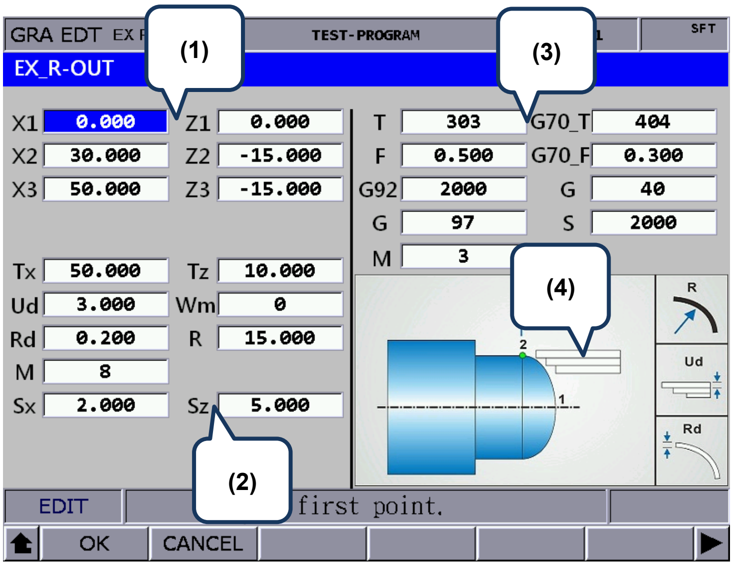
- The input values for Wd, Qd, and Rd must be greater than 0.
- Cm is for setting the cutting mode. Set Cm to 0 to have the system perform peck turning. Set Cm to 1 to perform straight cutting. Set Cm to 2 to perform finish turning once according to the given path.

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

■ Outer convex turning [EX_ROUT]



(1) Input fields for coordinates of each step point

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
3rd point	X3_, Z3_	-	-

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning in X-axis direction	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Rd_	Allowance of finish turning	R_	Arc radius
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

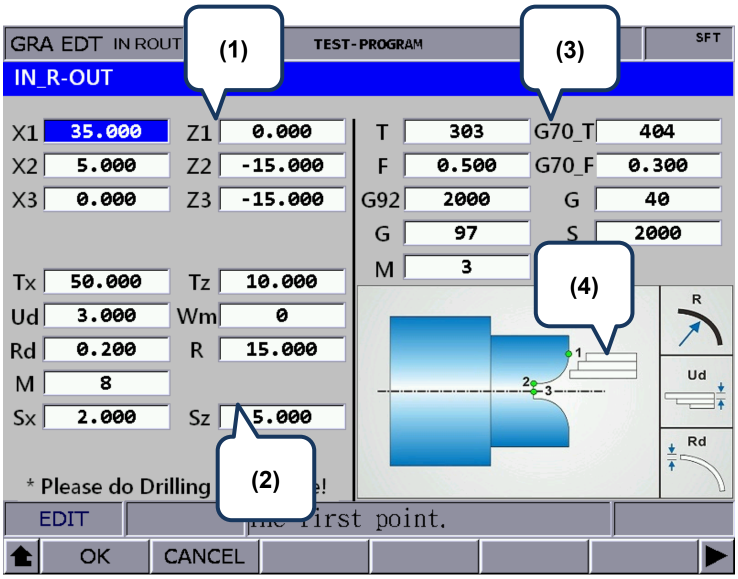
- R specifies the arc radius. Set R according to the X1-, Z1-, X2-, and Z2- coordinates to ensure the center of the arc and circle is the same. If the R value is inappropriate, an alarm occurs when you press **CYCLE START**.
- Wm is for setting the cutting mode. Set Wm to 0 to perform a complete turning process including rough and finish turning. Set Wm to 1 to perform rough turning and keep the allowance of finish turning. Set Wm to 2 to perform finish turning once according to the given path.

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

■ **Inner convex turning [IN_ROUT]**



(1) Input fields for coordinates of each step point

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
3rd point	X3_, Z3_	-	-

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning in X-axis direction	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Rd_	Allowance of finish turning	R_	Arc radius
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

■ R specifies the arc radius. Set R according to the X1-, Z1-, X2-, and Z2- coordinates to ensure the center of the arc and circle is the same. If the R value is inappropriate, an alarm occurs when you press **CYCLE START**.

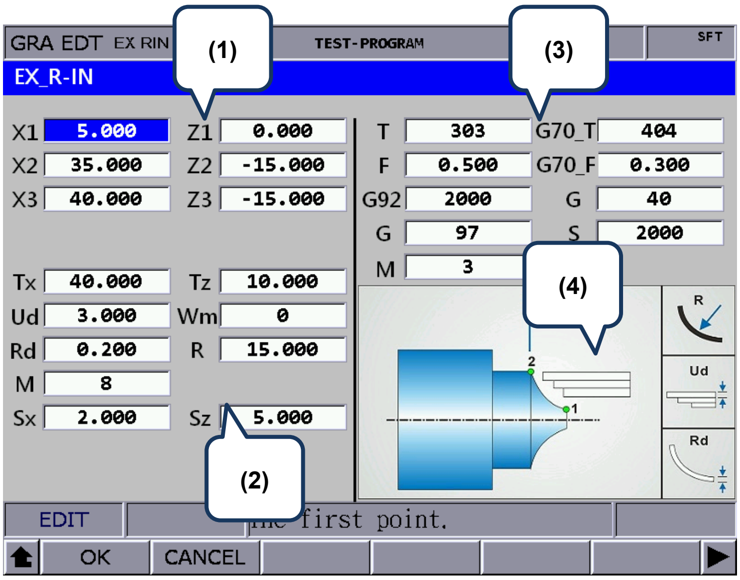
■ Wm is for setting the cutting mode. Set Wm to 0 to perform a complete turning process including rough and finish turning. Set Wm to 1 to perform rough turning and keep the allowance of finish turning. Set Wm to 2 to perform finish turning once according to the given path.

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

■ **Outer concave turning [EX_RIN]**



(1) Input fields for coordinates of each step point

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
3rd point	X3_, Z3_	-	-

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning in X-axis direction	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Rd_	Allowance of finish turning	R_	Arc radius
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

■ R specifies the arc radius. Set R according to the X1-, Z1-, X2-, and Z2- coordinates to ensure the center of the arc and circle is the same. If the R value is inappropriate, an alarm occurs when you press **CYCLE START**.

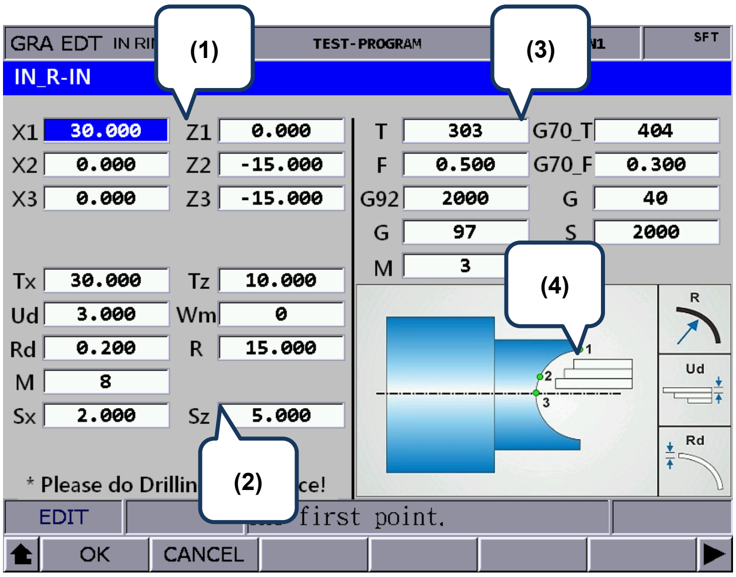
■ Wm is for setting the cutting mode. Set Wm to 0 to perform a complete turning process including rough and finish turning. Set Wm to 1 to perform rough turning and keep the allowance of finish turning. Set Wm to 2 to perform finish turning once according to the given path.

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

■ **Inner concave turning [IN_RIN]**



(1) Input fields for coordinates of each step point

Item	Description	Item	Description
1st point	X1_, Z1_	2nd point	X2_, Z2_
3rd point	X3_, Z3_	-	-

(2) Parameter settings for contour machining

Item	Description	Item	Description
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning in X-axis direction	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Rd_	Allowance of finish turning	R_	Arc radius
M_	Coolant switch (8 = on; 9 = off)	Sx, Sz	Safety clearance (coordinates)

- R specifies the arc radius. Set R according to the X1-, Z1-, X2-, and Z2- coordinates to ensure the center of the arc and circle is the same. If the R value is inappropriate, an alarm occurs when you press **CYCLE START**.

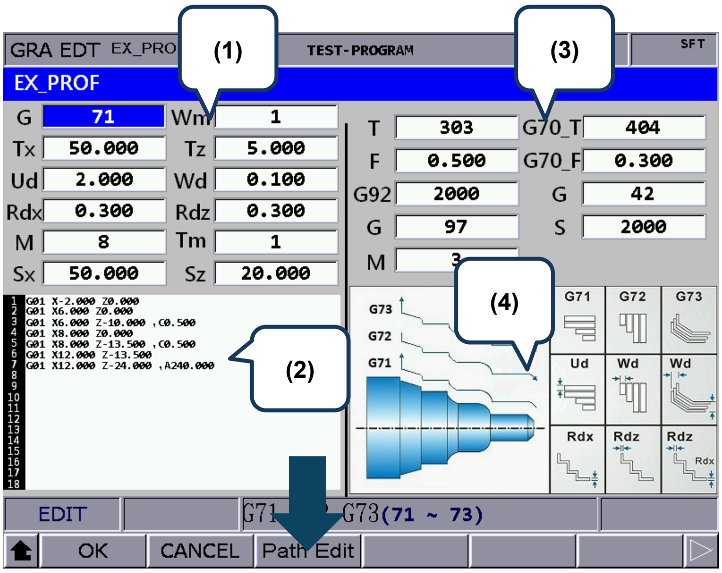
- Wm is for setting the cutting mode. Set Wm to 0 to perform a complete turning process including rough and finish turning. Set Wm to 1 to perform rough turning and keep the allowance of finish turning. Set Wm to 2 to perform finish turning once according to the given path.

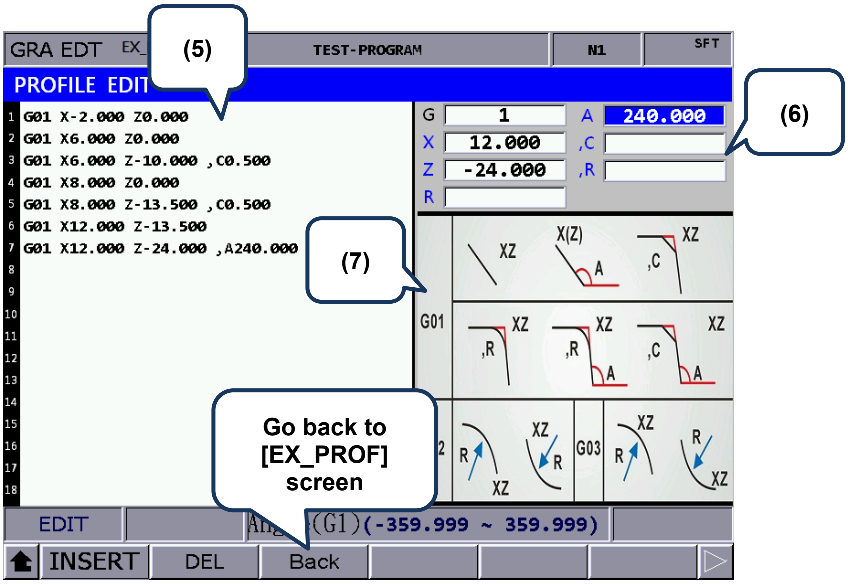
(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

■ **External profile turning [EX_PROF]**





(1) Settings of machining parameters

Item	Description	Item	Description
G_	Rough turning cycle selection	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning (X-axis direction)	Wd_	Feeding amount of rough turning (Z-axis direction)
Rdx_	Allowance of finish turning (X-axis direction)	Rdz_	Allowance of finish turning (Z-axis direction)
M_	Coolant switch (8 = on; 9 = off)	Tm_	Number of cutting in a cycle
Sx, Sz	Safety clearance (coordinates)	-	-

- G71 / G72 / G73 are for setting the profile turning type.

  G71 is suitable for machining of longer and thinner bars in Z-axis direction.
  G72 is suitable for machining of shorter and thicker bars in X-axis direction.
  G73 is suitable for workpiece that has been machined (such as forging or roughing).

- Ud is available for G71 or G73 command. It specifies the feeding amount of each rough turning in X-axis direction in a G71 command and specifies the total cutting depth in X-axis direction in a G73 command.

- Wd is available for G72 or G73 command. It specifies the feeding amount of each rough turning in Z-axis direction in a G72 command and specifies the total cutting depth in Z-axis in a G73 command.

- Tm is only available for G73 command, specifying the number of cutting in a cycle. The feeding amount of each cut is dividing the total feeding amount specified in Ud and Wd by Tm.

(2) Profile edit screen

The program generated in the PROFILE EDIT screen is displayed in this function. Or you can directly enter the program in this section by using ↑ and ↓ to move the cursor to the line to be edited.

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

(5) Profile edit screen. The function is the same as that of section (2).

(6) Program edit screen

Enter the parameters of the G-code in this section and press **INSERT** to insert the program.

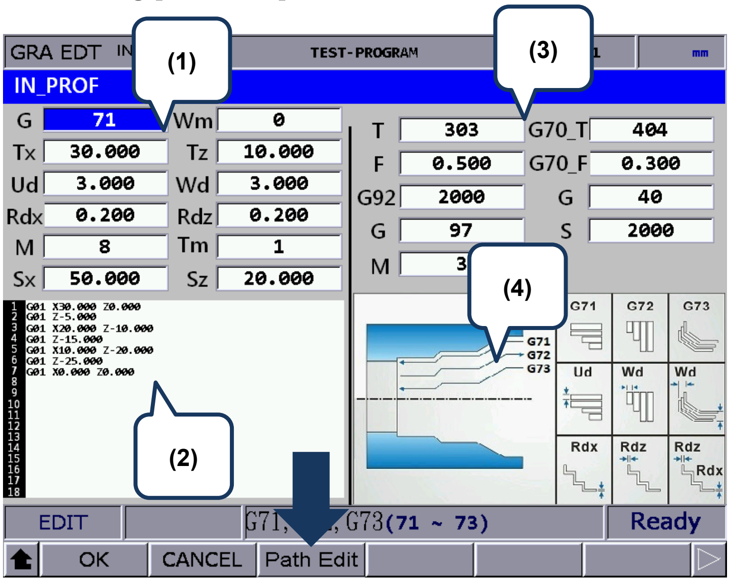
(7) Illustration for each command format

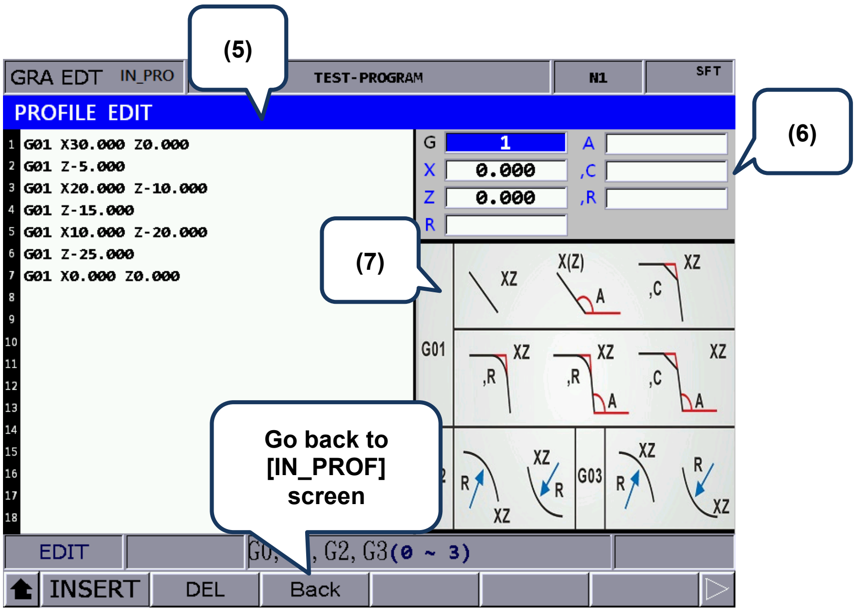
When editing the path, you can create the path according to the command formats shown in the above figure. The three types in this section are G01, G02, and G03.

There are nine formats for the G01 command, including G01X_Z_, G01X_,A_, G01Z_,A_, G01X_Z_,C_, G01X_Z_,R_, G01X_,A_,R_, G01Z_,A_,R_, G01X_,A_,C_, and G01Z_,A_,C_.

The G02 format is G02X_Z_R_ and the G03 format is G03X_Z_R_. ",C" represents automatic chamfer and ",R" represents automatic corner rounding.

■ Internal profile turning [IN_PROF]





(1) Settings of machining parameters

Item	Description	Item	Description
G_	Rough turning cycle selection	Wm_	Cutting mode setting (0 = rough & finish; 1 = rough; 2 = finish)
Tx_	X-coordinate of tool change position	Tz_	Z-coordinate of tool change position
Ud_	Feeding amount of rough turning (X-axis direction)	Wd_	Feeding amount of rough turning (Z-axis direction)
Rdx_	Allowance of finish turning (X-axis direction)	Rdz_	Allowance of finish turning (Z-axis direction)
M_	Coolant switch (8 = on; 9 = off)	Tm_	Number of cutting in a cycle
Sx, Sz	Safety clearance (coordinates)	-	-

- G71 / G72 / G73 are for setting the profile turning type.

  G71 is suitable for machining of longer and thinner bars in Z-axis direction.
  G72 is suitable for machining of shorter and thicker bars in X-axis direction.
  G73 is suitable for workpiece that has been machined (such as forging or roughing).
- Ud is available for G71 or G73 command. It specifies the feeding amount of each rough turning in X-axis direction in a G71 command and specifies the total cutting depth in X-axis direction in a G73 command.
- Wd is available for G72 or G73 command. It specifies the feeding amount of each rough turning in Z-axis direction in a G72 command and specifies the total cutting depth in Z-axis in a G73 command.
- Tm is only available for G73 command, specifying the number of cutting in a cycle. The feeding amount of each cut is dividing the total feeding amount specified in Ud and Wd by Tm.

(2) Profile edit screen

The program generated in the PROFILE EDIT screen is displayed in this function. Or you can directly enter the program in this section by using and to move the cursor to the line to be edited.

(3) Parameter settings for tool compensation, tool nose radius compensation, speed, and feedrate

Refer to [EX_STEP] for the parameter settings in section (3).

(4) Illustration

(5) Profile edit screen. The function is the same as that of section (2).

(6) Program edit screen

Enter the parameters of the G-code in this section and press **INSERT** to insert the program.

(7) Illustration for each command format

When editing the path, you can create the path according to the command formats shown in the above figure. The three types in this section are G01, G02, and G03.

There are nine formats for the G01 command, including G01X_Z_, G01X_,A_, G01Z_,A_, G01X_Z_,C_, G01X_Z_,R_, G01X_,A_,R_, G01Z_,A_,R_, G01X_,A_,C_, and G01Z_,A_,C_.

The G02 format is G02X_Z_R_ and the G03 format is G03X_Z_R_. ",C" represents automatic chamfer and ",R" represents automatic corner rounding.

7.15.4 Operation steps for graphic programming

(1) Set the controller to EDIT mode to enter the Graphic edit (GRA EDT) screen.

(2) **GRA EDT** is on the last page of the function bar in the File manage screen.

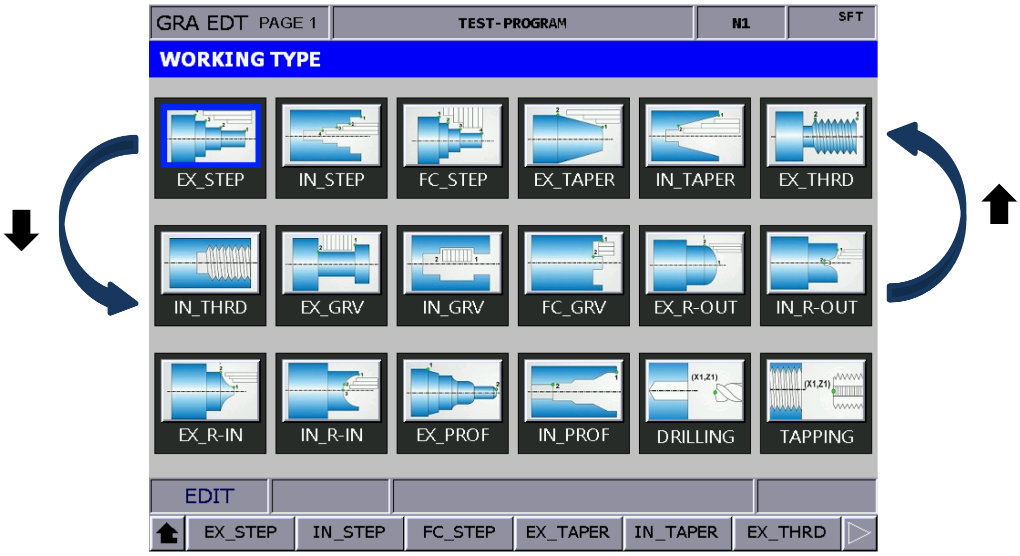
(3) Enter the Graphic edit screen and then select the disk type (CF or USB).

(4) After entering the disk, press **ADD PRJ**, enter the file name, and press **ENTER** to generate a project file.

(5) Move the cursor to one of the project files and press **ENTER** to enter the file.

(6) After entering the project file, press **ADD PRC** to enter the screen of procedure selection.

(7) There are 18 procedures available. Every 6 procedures are in a row, respectively corresponding to the function keys. Press ⬆ and ⬇ to shift the cursor to another row. When you shift to another row of procedures, the screen of the corresponding function keys changes as well.



(8) Select the required procedure and press the corresponding function key to enter the screen for procedure editing.

(9) After setting the parameters, press **OK**, name the procedure, and press **ENTER** to finish the editing.

(10) If desiring to continue with other procedures, press **ADD PRC** again.

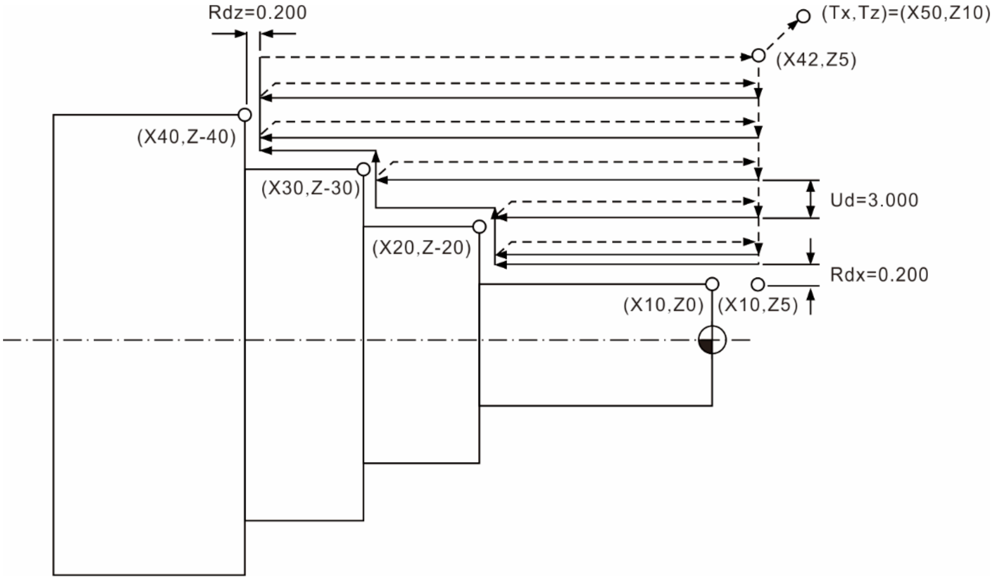
(11) After finishing editing the procedures, press **CONVERT** to convert the files into G-code format.

(12) The system jumps to the File edit screen after file conversion, and you can verify the program in the screen. Then, set the system to GRA (graphic display) mode, press **PREVIEW** to ensure the generated graph is correct, and then you can perform MPG simulation or actual machining.

7

[File conversion example_External step turning]

The following illustrates the G-code file generated by the parameters for the external step turning.



O0000 // Character string set in the INITIAL file, which is automatically added to the beginning of the program.
;EXstep_DR0
T0303 //Tool number and compensation number
G95 F0.500 //0.5 mm/feeding amount per revolution
G92 S2000 //Maximum spindle speed
G97 S1000 //Fixed spindle speed
M3 //Spindle ON - clockwise
M8 //Coolant ON
G00 X42.000 Z5.000
G42 //Tool nose radius compensation ON
G71 U3.000 R0.5 //Feeding amount of rough turning in X-axis direction
G71 P210 Q250 U0.200 W0.200 //Allowance of finishing turning in X- and Z-axis directions
N210 G00 X10.000 Z5.000
G01 X10.000 Z0.000 //1st point
G01 Z-20.000
G01 X20.000 Z-20.000 //2nd point
G01 Z-30.000
G01 X30.000 Z-30.000 //3rd point
G01 Z-40.000
G01 X40.000 Z-40.000 //4th point
N250 G01 U2.0 //U2.0
G00 X42.000 Z5.000
T0404 //Tool for finish turning
G70 P210 Q250 F0.100 //Program block for finish turning
G00 Z5.000
G40
G00 X50.000 Z10.000 //Coordinates of tool change point
M09
M05
M30 // Word string in the **FINISH** file, which is automatically added to the end of the program

7.16 Program function in other modes

AUTO mode:

The PRG screen displays the content of the opened G-code file. In the screen, you can view the status information about the opened or executed file and the block being executed. The PRG screen in AUTO mode displays information about the program and the coordinates of motion trajectory during program execution. The operation steps are as follows.

(1) In AUTO mode, press **PRG** to display the program execution screen, as shown in the following figure.

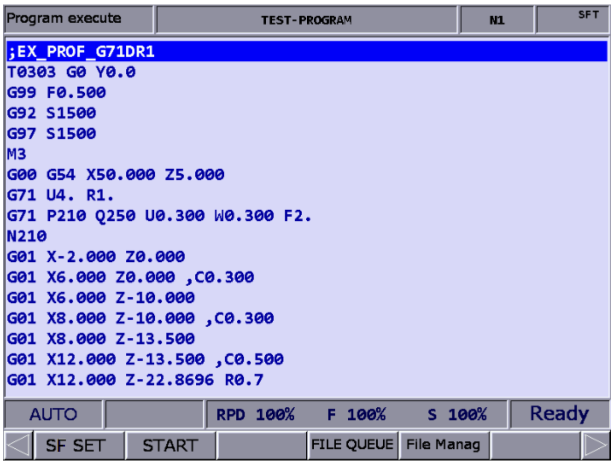


Figure 7.16.1

(2) Continued from Step (1), press **PRG** again, and the screen displays the information of program content and coordinates simultaneously, as shown in the following figure.

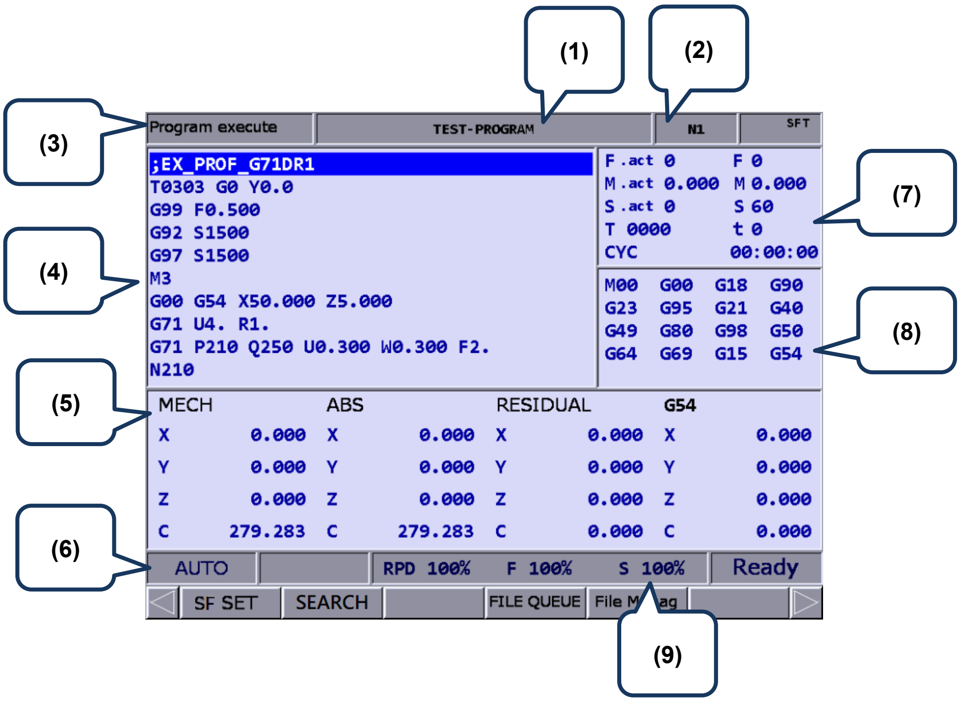


Figure 7.16.2

(1) Name of current program

(2) Currently executed program line

(3) Current group function

(4) Currently executed program content

(5) Coordinate information during program execution

(6) Current system mode

(7) F.act: actual feedrate

S.act: actual spindle speed

D: tool radius compensation number

H: tool length compensation number

T: tool number

F: feedrate

S: spindle speed

t: dwell time

CYC: cycle time

(8) Current command status

(9) Current override settings

The function of break line search is available in AUTO mode. When the program execution is interrupted, the system records the line number where it is interrupted (break line). You can go to the PRG screen in AUTO mode to enable the break line search function.
When the system searches the break line, the cursor quickly moves to the line/label number you searched for and the system quickly computes and executes the program before the specified block to ensure the machining status is ready (including the spindle speed, feedrate, M code, and coordinates) when the execution resumes, as shown in the following figure.

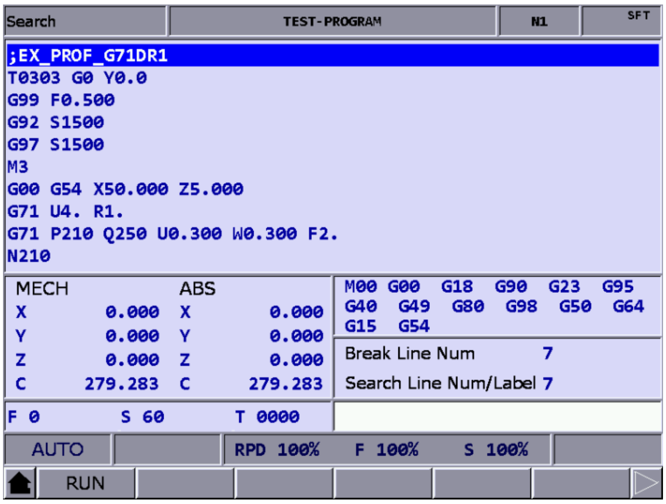


Figure 7.16.3

The operation steps are as follows.

(1) In AUTO mode, press **PRG** to enter the program execution screen.

(2) Press **SEARCH** to display the Search screen.

(3) Refer to the information of break line number, enter the line or label number of the program to be searched, and then press **ENTER** to complete the setting.

(4) Press **RUN**, the system executes the program until reaching the specified line or label of the program.

(5) The controller executes and records the execution status of the program blocks prior to the specified line. Then the controller stops at the break line for execution.

(6) Press **CYCLE START** to execute the program.

Note:

1. When finding the target block, the system stops and remains unexecuted. Press **CYCLE START** to resume executing the program.
2. Supported formats for searching: line number and label (N number) of the program.
3. During program execution or the break line search function is used, any request for break line search will be ignored as the system regards it is in execution.

7

When the G-code program is executed, you can use the SF set function to change the cutting feedrate (F command) and spindle speed (S command) specified in the G-code program, as shown in Figure 7.16.4. Enter a new command value in the SF set dialog box to change the speed command during execution.

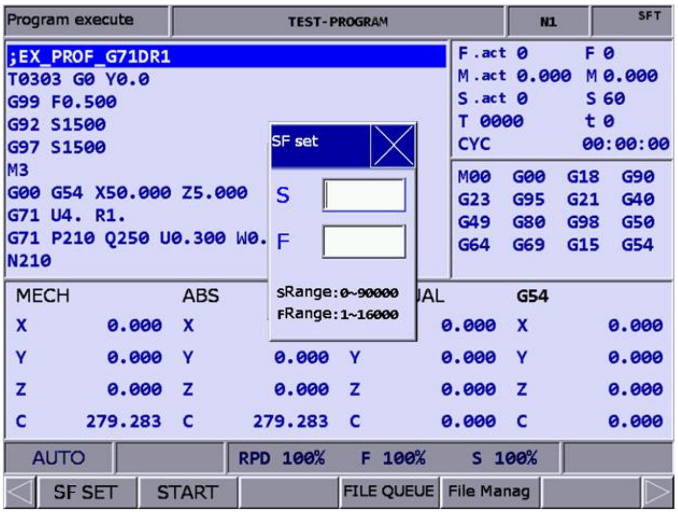


Figure 7.16.4

The operation steps are as follows.

(1) In AUTO mode, press **PRG** to enter the program execution screen.

(2) Press **SF SET** and a dialog box appears for you to enter the command value.

(3) Enter the S value or F value and press **ENTER** to complete changing the speed command setting.

Note:

1. The SF set function is a one-shot function with the S and F settings effective for one time in a single execution, which do not change the commands in the G-code program. If requiring to execute this SF setting for multiple times, you are suggested to edit the command in EDIT mode to ensure the speed command is correct.
2. After the S value is set, the current spindle speed in the G-code program is changed immediately. After the F value is set, the system executes with the new feedrate (F command) after the data in the system buffer is completely processed.
3. If there is no S or F command in the G-code program, you cannot use this function to change the speed command.
4. When using the SF set function, set Pr10017 [Bit 3] SF speed setting to enable or disable the F setting.

The Barcode reader function is for using the barcode scanner to load the machining files named by barcode into the file queue and execute them, which greatly saves the time for file searching. You can connect the barcode reader to the USB slot in the front side of the controller.

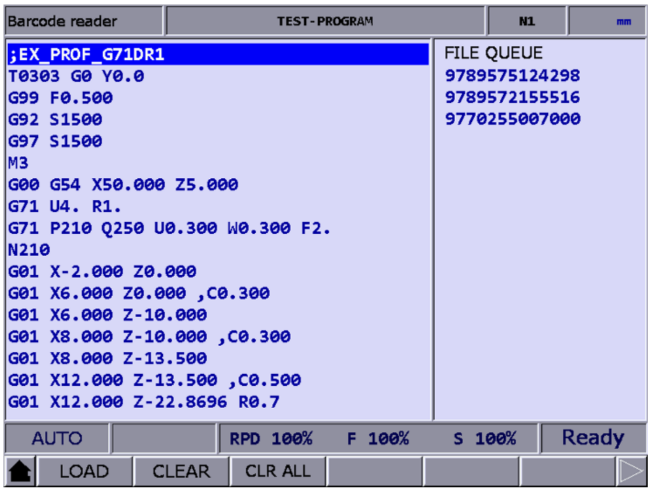


Figure 7.16.5

The operation steps are as follows.

(1) In AUTO mode, press **PRG** to enter the program execution screen.

(2) Press **BARCODE** to enter the screen as shown in Figure 7.16.5.

(3) Use the barcode scanner to scan the barcode to obtain the name of the machining file.

(4) To load the file content, press **LOAD**. To delete the file which is on the top of the FILE QUEUE list, press **CLEAR**. To delete all the files on the FILE QUEUE list, press **CLR ALL**.

Note:

1. To use this function, you must first create the machining files in the CF card and the file names have to be consistent with their barcodes.
2. When loading multiple files, the system executes each file in sequence. It automatically deletes the file name from the list once completing the execution.

JOG and MPG modes:

The operation steps for SF setting are as follows.

(1) In JOG or MPG mode, press **PRG** to enter the program execution screen.

(2) Press **SF SET** and a dialog box appears for you to enter the command value.

(3) Enter the S value or F value and press **ENTER** to complete changing the speed command setting.

With the TEACH function, you can manually move the axis to any position and use the programming function keys, and the system can convert the final position (coordinates of the three axes) into a motion block with coordinates specified. Set the system to JOG or MPG mode, and you can use the TEACH function in the PRG screen for programing the existing or new files. Functions in the TEACH screen include: rapid traverse (POSITION), linear interpolation (LINEAR), circular interpolation (CIRCULAR), delete (DEL), file creating (NEW FILE), file saving (SAVE), and absolute / machine coordinates (ABS / MECH). This programming function automatically generates corresponding command formats according to the different functions. See the following table for the generated command formats.

Function	Auto-generated command format
NEW FILE (generate a new file when teach programming is enabled)	G90 G40 G49 G98 G50 G64 G80 G17 G69 G21 G54 G15 S3000 M03 F1000 Note: this function generates the corresponding G21 or G20 command according to the parameter setting of size unit.
POSITION (rapid traverse)	G00 + X_Y_Z_
LINEAR (linear interpolation)	G01 + X_Y_Z_
CIRCULAR (circular interpolation)	G02 + X_Y_Z_ + I_ J_ or G03 + X_Y_Z_ + I_ J_ Note: this function generates G17+I_ J_, G18+K_ I_, or G19+J_K_ corresponding to the X-Y, Z-X, or Y-Z plane.
ABS (absolute coordinates)	G90 G00 (or G01 / G02 / G03) + X_Y_Z_
MECH (machine coordinates)	G53 G00 (or G01 / G02 / G03) + X_Y_Z_

The operation steps for the TEACH function are as follows.

(1) In JOG or MPG mode, press **PRG** to enter the program execution screen.

(2) Press **TEACH** to enter the teach screen.

(3) Select the file to be programmed from the existing files or create a new file. If desiring to do programming in the existing file, open the file in EDIT mode. If desiring to do programming in a new file, press **NEW FILE**, and a dialog box appears for you to enter the file name. Enter the file name, press **ENTER**, and you can create a new file in the current directory.

(4) Specify the data type of coordinates. For example, if you desire to display absolute coordinates, press **ABS** on the second page of the function bar. Or press **MECH** to switch to machine coordinates.

(5) Move the axis to the specified position in JOG or MPG mode, press **POSITION** or **LINEAR** according to the requirement of the mode, and the coordinate command is inserted to the position where the cursor is located. The coordinate command is generated according to the data type of the coordinate values.

(6) Continue from Step (5), to specify a circular motion, press **CIRCULAR** to display the corresponding function bar.

(7) Continue from Step (6), to specify the plane of the arc, press **PLANE** to select X-Y, Y-Z, or Z-X.

(8) Move the axes in sequence and set the start, intermediate, and end points of the arc by pressing **P1**, **P2**, and **P3** respectively. After P3 is set, the values are automatically converted into a circular cutting command. The system determines the direction of the arc (G02 or G03) and the radius according to the trajectory of P1 - P3.

(9) If the coordinate command is incorrect, move the cursor to the block and press **DEL** (on the function bar of the 1st layer in the teach screen) to delete the block.

(10) After completing the programming, in addition to the given auto-saving mechanism (press **RESET**, switch to different system modes, or open another file), you can press **SAVE** to save the programming results.

Note:

1. The Teach function is only available in JOG or MPG mode. It is not displayed in other modes.
2. The allowable file size for the teach function is the same as that for file editing (below 3 MB).
3. The name of the created file for the teach function must comply with the file name specification.
4. If you repeatedly enter two sets of coordinate with the same values, the system ignores the 2nd coordinate command to avoid generating an invalid motion block.
5. Set the coordinates of P1, P2, and P3 for the arc command in sequence. The direction and radius for the circular command is determined by the positions of P1, P2, and P3.
6. If you enter the teach screen without opening a file, the system automatically generates a blank file named “TEACH.NC” in the directory where the cursor is located (the default setting is to generate a file in the root directory of CF), so you can directly use the teach function.
7. When using the SF set function, set Pr10017 [Bit 3] SF speed setting to enable or disable the F setting.

MDI mode:

In MDI mode, you can enter simple programs and save, delete, or execute the content in the PRG screen, as shown in the following figure. You can enter up to 14 program blocks in the screen. After finishing editing the program, press **LOAD** to reload and then execute the program. Otherwise, the program cannot be executed.

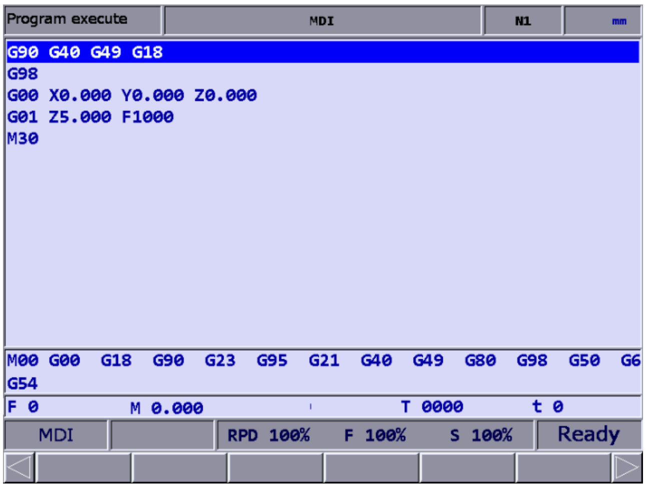


Figure 7.16.6

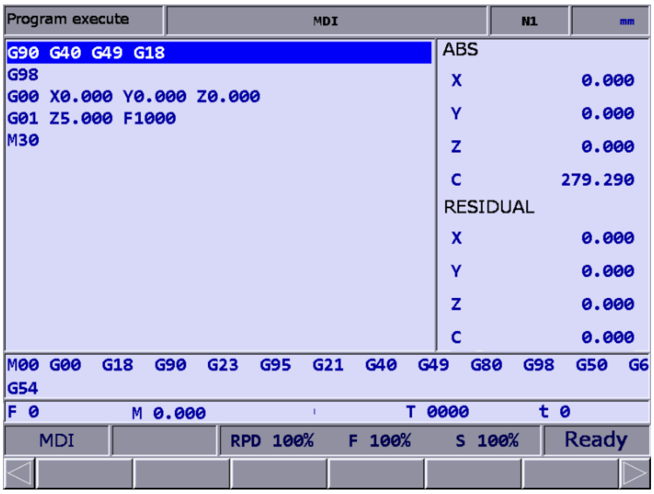


Figure 7.16.7

In addition, the SAVE function is for saving the program content entered in MDI mode as a file in the current directory. The naming method is the same as that in Section 7.2 Create a new file. That is, the file name has to follow the naming convention and be unique. The CLR function is for deleting all the program content in the screen in MDI mode, which functions the same as pressing and holding **RESET** (you can press and hold **RESET** for 3 seconds to delete all the content in the program screen).

Note:

1. In MDI mode, the **RESET** key has a two-stage function. Press **RESET** once to return to the first line of the manual input program after program interruption. Press and hold **RESET** for 3 seconds to clear all the manual input program contents.
2. In MDI mode, after M30 is executed, the cursor returns to the first line of the program. You can resume the execution without reloading the program.
3. In MDI mode, if there is no M30 (Program end) command, the program runs to the last line. To resume the execution, press **LOAD** to reload the program.
4. In MDI mode, after M02 is executed, the cursor stops at the block of M02 and the program status restores to the default. And you can resume the execution from the block of M02 without reloading the program.

(This page is intentionally left blank.)

Offset (OFS) group

8

The OFS group provides functions for setting the workpiece coordinates, tool length, tool radius compensation, and macro variables.

The OFS group provides functions for setting the workpiece coordinates, tool length, tool radius compensation, and macro variables.

Note: bold function names in a box (such as **POS**) mean the keys on machine operation panel A;
bold function names (such as **CLR ALL**) mean the function keys of F1 - F6.

8.1 Coordinate setting (COORD)

G54 - G59 allow you to set multiple workpiece coordinate systems. With commands G54 - G59, you can simplify the calculation of coordinates during programming as well as change the coordinate values at any time, achieving more flexible machining process. You can specify the coordinate values in the coordinate setting screen with one of the workpiece coordinate commands (G54 - G59), as shown in the following figure.

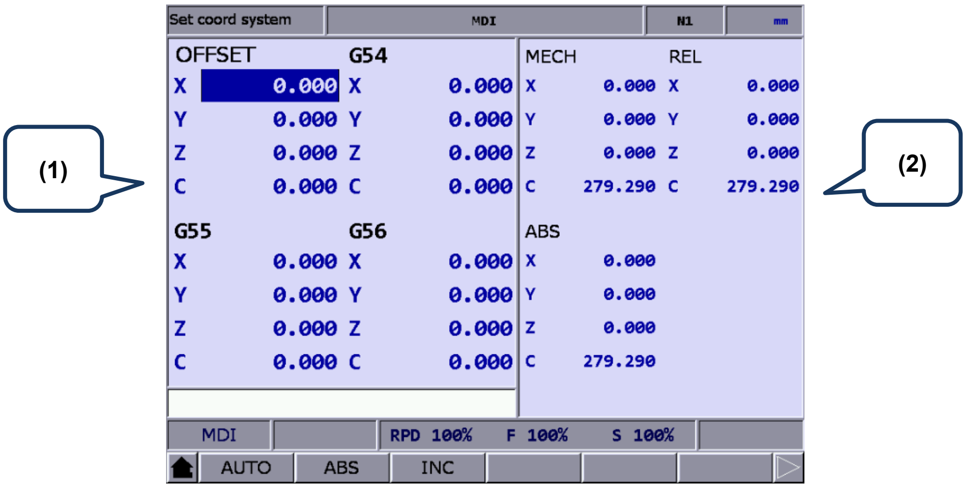


Figure 8.1.1

(1) Workpiece coordinate setting: offset coordinates, G54 - G59 coordinate systems

(2) Coordinate information: machine (MECH) / relative (REL) / absolute (ABS) coordinates

The operation steps are as follows.

(1) Press **OFS** to enter the OFS screen.

(2) Press **COORD** to display the corresponding function bar.

Note:

1. Set the coordinate systems only when there is no machining program being executed. Otherwise, data entry is prohibited.
2. If you press **FEED HOLD** during program execution, the system status remains “RUN”. If you press (Single block execution) during program execution, the execution stops after the current block is finished, and the system status is “Ready.”.

8.1.1 Auto set (AUTO)

This function is for automatically entering the current position of each axis to the coordinate system (G54 - G59) where the cursor is located. The auto set function includes three options: setting single axis (SET), setting multiple axes (SET P), and setting coordinate system center (SET L/2). The option of setting the coordinate system center must be used with the function of clearing relative coordinates (CLR REL). The function of clearing coordinate values of a coordinate system (CLR ALL) is also provided.

- CLR ALL (all clear): clears all the axis values of the current coordinate system to 0 while the values in other systems remain unchanged.
  The operation steps are as follows.
  (1) Press **OFS** to enter the OFS screen.
  (2) Press **COORD** to display the corresponding function bar.
  (3) Press **AUTO** to display the corresponding function bar.
  (4) Press ↑, ↓, ←, and → to move the cursor to a specific coordinate system.
  (5) Press **CLR ALL** to delete all the data of the coordinate system.

- CLR REL (relative clear): clears the relative coordinates of the axis which the cursor is pointing to. This function is not used for clearing the actual workpiece coordinates but for clearing the displayed relative coordinates.

- SET L/2 (set center): this function is for setting the central position of an object as the center of a coordinate system. The NC system automatically calculates and enters the central position coordinate to the field, so you do not need to do it manually. The following operation steps take the X axis as an example.
  (1) Set the system to JOG or MPG mode and move the machine axis to the initial contact point of the workpiece in X-axis direction.
  (2) Press **OFS** to enter the OFS screen.
  (3) Press **COORD** to display the corresponding function bar.
  (4) Press **AUTO** to display the corresponding function bar.
  (5) Press ↑, ↓, ←, and → to move the cursor to the X-coordinate field of a specific coordinate system.
  (6) Press **SET L/2** to enter its setting screen.

(7) Press **Point1** and the circle on the left side of the rectangle becomes red, as shown in Figure 8.1.1.1, meaning the machine coordinates of the first point is recorded.

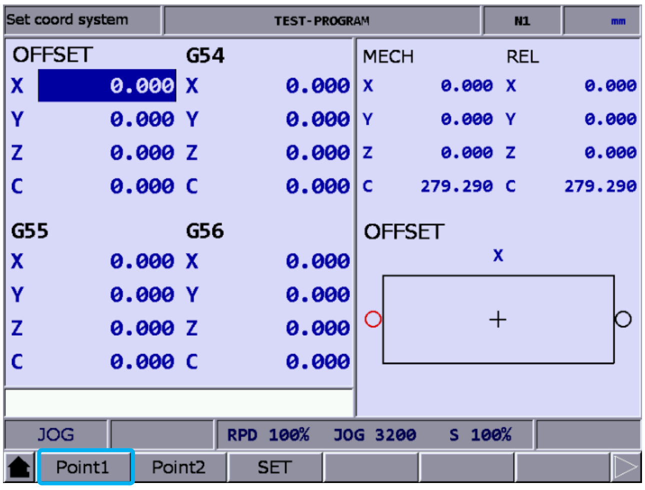


Figure 8.1.1.1

(8) Continue to move the machine axis to the other contact point of the workpiece in X-axis direction.

(9) Press **Point2** and the circle on the right side of the rectangle becomes red, meaning the machine coordinates of the second point is recorded.

(10) Press **SET** and the system automatically calculates the central point between the machine origin and the workpiece position in X-axis direction and sets this point as the center of X axis in the coordinate system, which is the workpiece origin of X axis.

- SET L (set single axis): this function is for automatically entering the current machine coordinate of a single axis. When you move the cursor to the X, Y, or Z field of a specific coordinate system and press **SET L**, the current machine coordinate is automatically entered to the field where the cursor is located. This function is only for entering the data of a single axis. The operation steps are as follows.

(1) Set the system to JOG or MPG mode and move the machine axis to the initial contact point of the workpiece in X-axis direction.

(2) Press **OFS** to enter the OFS screen.

(3) Press **COORD** to display the corresponding function bar.

(4) Press **AUTO** to display the corresponding function bar.

(5) Press , , , and to move the cursor to the X-coordinate field of a specific coordinate system.

(6) Press **SET L** to automatically enter the axis coordinate value in the field where the cursor is located.

Example of setting single axis

This example illustrates setting X-axis value by moving the machine axis to a specific position (workpiece origin in X-axis direction as shown in Figure 8.1.1.2).

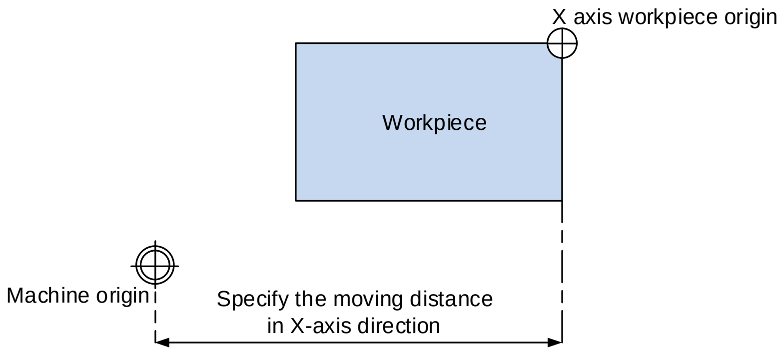


Figure 8.1.1.2

Then the coordinate values are displayed in the machine coordinate fields as shown in Figure 8.1.1.3. Move the cursor to a specific coordinate system (such as G54 as shown in Figure 8.1.1.3) and press **SET L** to automatically enter the X-axis value of the machine coordinate to the X-axis field in G54 coordinate system, completing the data entry for single axis.

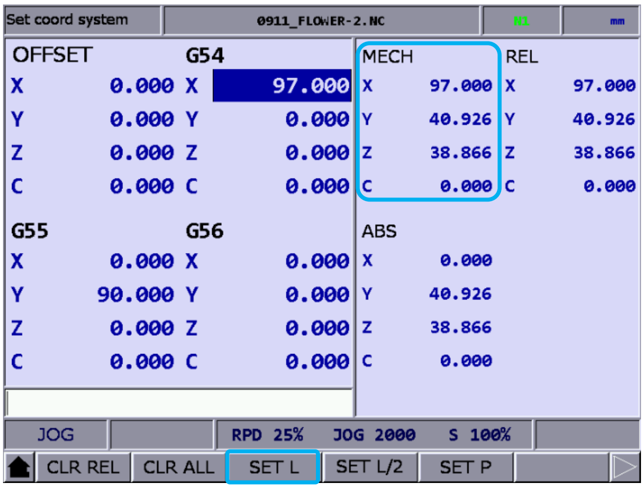


Figure 8.1.1.3

- SET P (set multiple axes): this function is for automatically entering the coordinates of multiple axes. After completing the calibration of workpiece center, you can use this function to enter the machine coordinates of multiple axes (including X, Y, Z, and other axes) simultaneously. The operation steps are as follows.
  (1) Set the system to JOG or MPG mode and move the machine axis to the initial contact point of the workpiece in X-axis direction.
  (2) Press **OFS** to enter the OFS screen.
  (3) Press **COORD** to display the corresponding function bar.
  (4) Press **AUTO** to display the corresponding function bar.
  (5) Press ↑, ↓, ←, and → to move the cursor to a specific coordinate system.
  (6) Press **SET P** to automatically enter the coordinate values of multiple axes in the coordinate system field where the cursor is located.

Note: if you have set the coordinates of other axes, do not press **CLR ALL** to clear the axis values, or the coordinate values are all cleared.

Example of setting multiple axes

Move the machine axis to a specific position as the workpiece origin shown in Figure 8.1.1.4 (the figure illustrates the position of X and Y axes except Z axis).

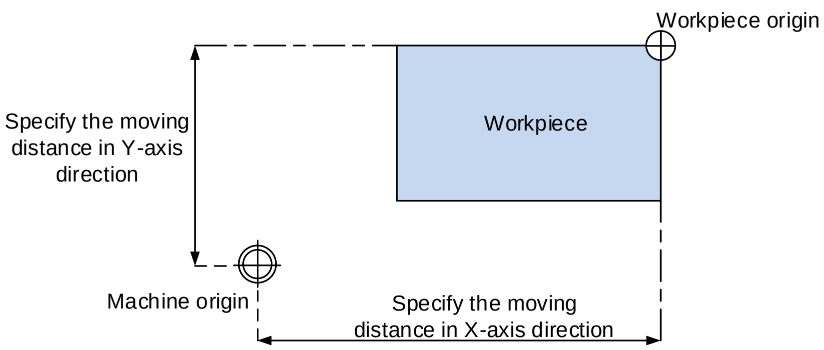


Figure 8.1.1.4

After completing the calibration of workpiece origin, the coordinate values are displayed in the machine coordinate fields as shown in Figure 8.1.1.5. Move the cursor to the G54 coordinate system and press **SET P** to automatically enter the values of X-, Y-, Z-, and C-axes of the machine coordinate to the X-, Y-, Z-, and C-axis fields in the G54 coordinate system, completing the data entry for multiple axes.

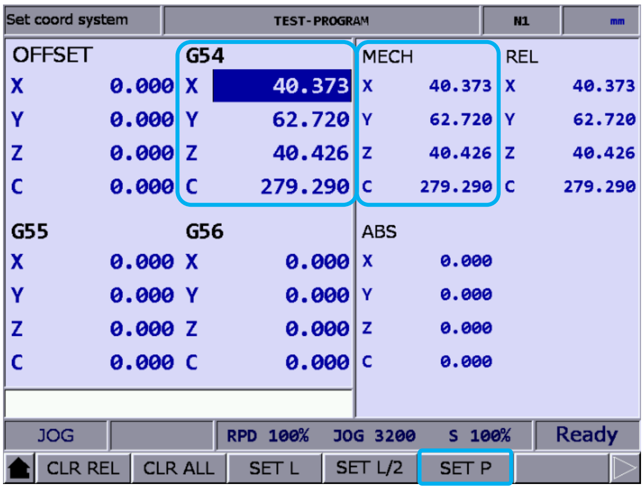


Figure 8.1.1.5

8.1.2 Absolute input (ABS)

One of the manual input function for coordinate values, which includes absolute and incremental settings. The following operation steps illustrate the absolute setting.

(1) Press **OFS** to enter the OFS screen.

(2) Press **COORD** to display the corresponding function bar.

(3) Press ↑, ↓, ←, and → to move the cursor to a specific coordinate system.

(4) To enter positive values, press **0** - **9**; to enter negative values, you have to press before using the numeral keys. After entering the values, press to determine the number of decimal places.

(5) Press **ABS** to enter absolute coordinates to the coordinate system.

Note:

1. The displayed values are in the unit of mm. If you enter values without specifying the decimal points, they are in the unit of μm. For example, “123456” = 123456 μm = 123.456 mm.

2. In Step (5), you can press either **ABS** or **ENTER** to enter the coordinates.

Example of absolute setting

Move the tool center from the machine origin to the workpiece origin (X, Y). Then, enter the X- and Y- machine coordinates corresponding to the workpiece origin for the coordinate setting (G54 - G59) in the OFS group.

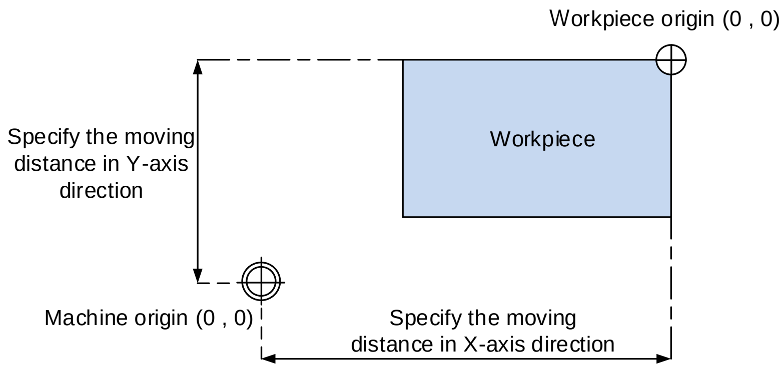


Figure 8.1.2.1

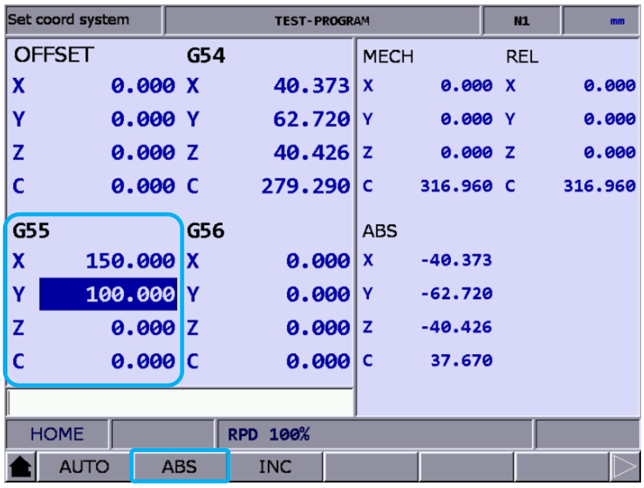


Figure 8.1.2.2

8.1.3 Incremental input (INC)

One of the manual input function for coordinate values, usually used for fine adjustment of the original value. That is, you change the values incrementally. For example, when the original value is 150.000, if you enter 5.000 by incremental setting, the newly-set value is 155.000.

The operation steps are as follows.

(1) Press **OFS** to enter the OFS screen.

(2) Press **COORD** to display the corresponding function bar.

(3) Press , , , and to move the cursor to the X-, Y, or Z-axis field of a specific coordinate system.

(4) To enter positive values, press **0** – **9**; to enter negative values, you have to press before using the numeral keys. After entering the values, press to determine the number of decimal places.

(5) Press **INC** to increment the coordinate values.

Note: make sure you use the correct mode (ABS or INC) and enter the correct coordinates to avoid danger caused by incorrect axis movement.

8

8.2 Tool setting

8.2.1 Tool length setting (Tool Offset)

- Functions of tool setting for the lathe system include tool length compensation, tool length wear compensation, tool nose radius, tool radius wear, and tool nose type. Prior to using the compensation functions, go to the tool setting screens in the OFS group to enter the tool compensation values. During program editing, specify the tool radius compensation number which corresponds to the number in the compensation data table.

- Format of tool compensation:

  T0204: 02 represents tool number 2 and 04 represents the tool length and tool wear compensation settings for tool number 4.

  T02: if only one set of number is specified, it means you use tool number 2 and its tool length and tool wear compensation settings for the machining, which is the same as T0202.

The tool length (Tool Offset) setting screen is as shown in the following figure.

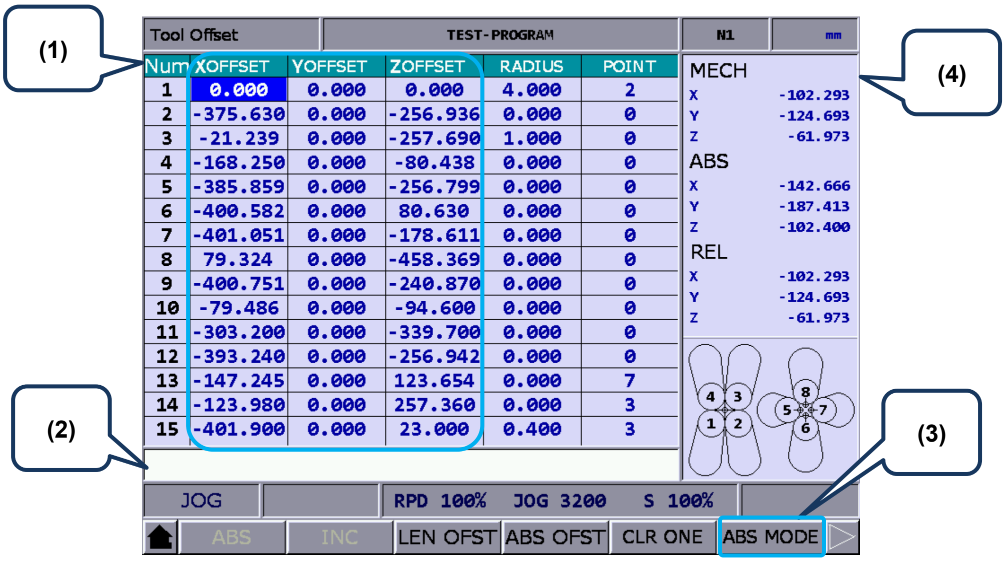


Figure 8.2.1.1

(1) Compensation number and tool length of the corresponding axis

(2) Input field for compensation data

(3) Input mode selection

(4) Auxiliary display: machine (MECH), absolute (ABS), and relative (REL) coordinates

Setting range for tool data	
Tool length	-2000.0 to 2000.0 mm

The tool wear setting screen is as shown in the following figure.

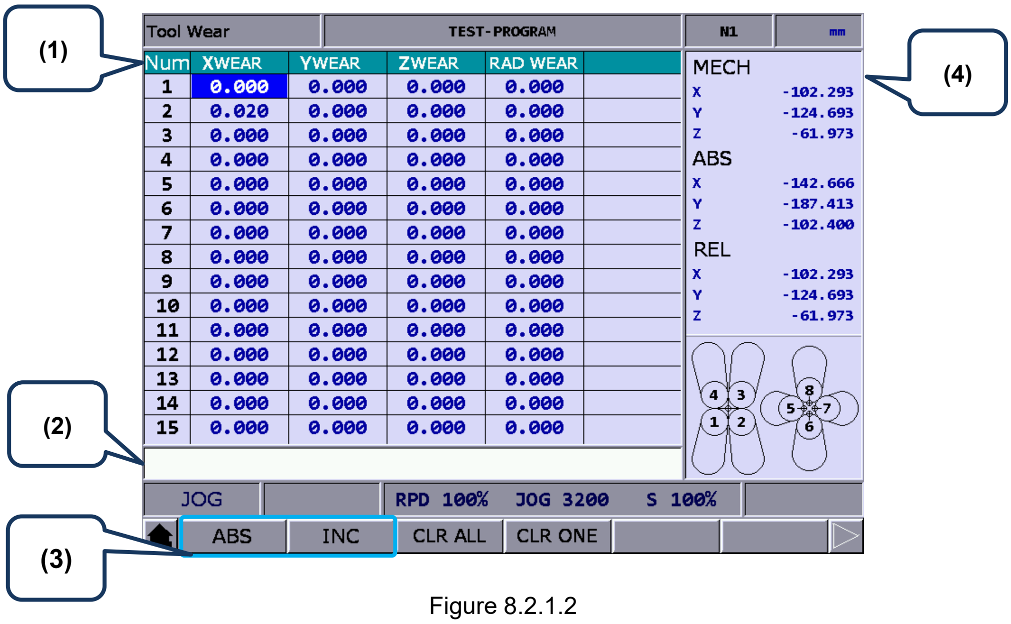


Figure 8.2.1.2

(1) Compensation number for tool wear and wear value of the corresponding axis

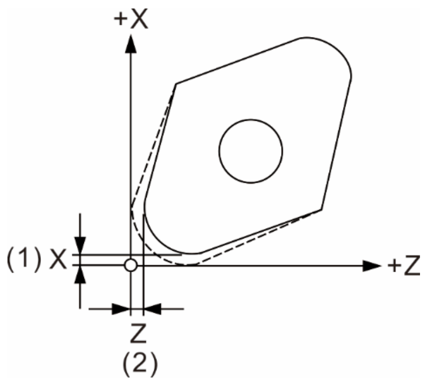
(2) Input field for compensation data

(3) Input mode selection

(4) Auxiliary display: machine (MECH), absolute (ABS), and relative (REL) coordinates

Setting range for tool data	
Tool wear for each axis	-2000.0 to 2000.0 mm
Tool radius wear compensation	-2000.0 to 2000.0 mm

Illustration of tool wear:



(1) Compensation amount of the tool nose wear for X axis

(2) Compensation amount of the tool nose wear for Z axis

The tool nose setting screen is as shown in the following figure.

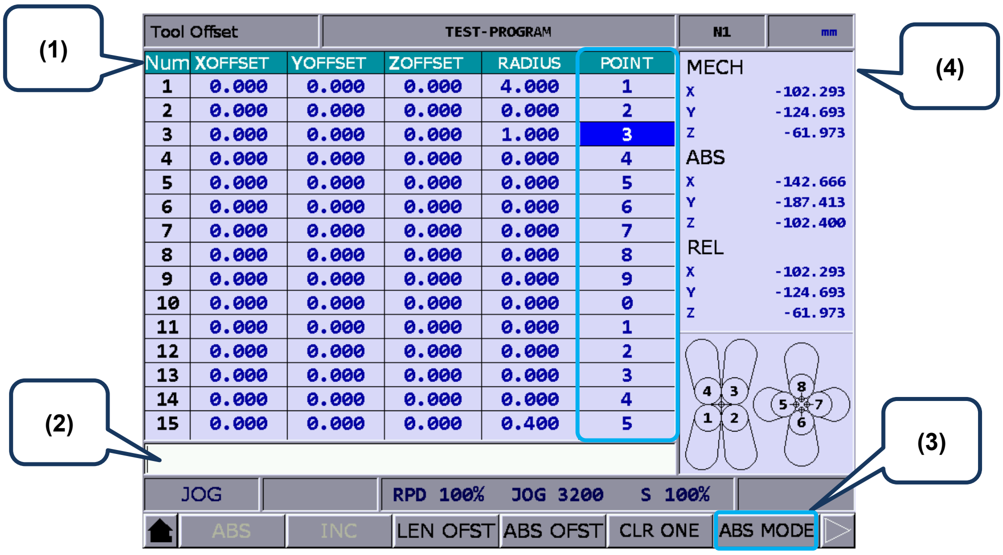


Figure 8.2.1.3

(1) Compensation number for tool nose radius and corresponding compensation value

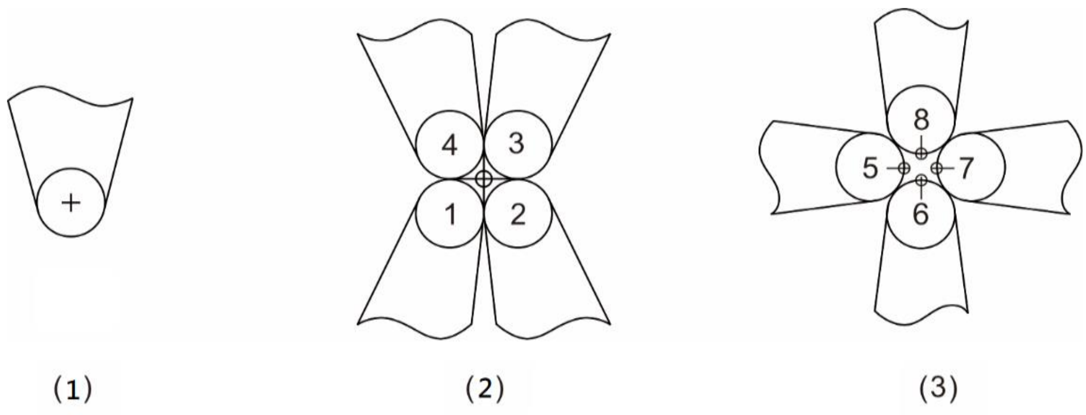
(2) Input field for compensation data

(3) Input mode selection

(4) Auxiliary display: machine (MECH), absolute (ABS), and relative (REL) coordinates

Setting range for tool data	
Tool nose radius compensation	-2000.0 to 2000.0 mm
Tool nose type	0 - 9

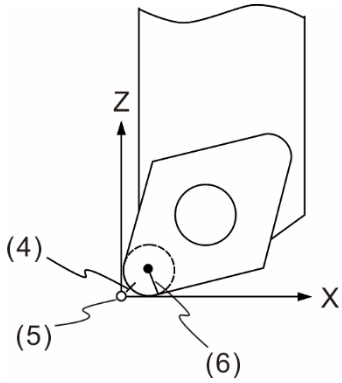
Tool nose types:



(1) Tool nose type 0 or 9 (2) Tool nose types 1 - 4 (3) Tool nose types 5 - 8

8

Illustration of tool nose position and tool nose radius compensation:



(4) Actual tool nose position

(5) Hypothetical tool nose position when calibration

(6) R value of tool nose radius compensation

Descriptions for Tool Offset function keys are as follows.

- ABS (absolute input): use this function to set absolute values for tool length, tool radius, tool wear compensation, or tool life. You can press either **ABS** or **ENTER** to enter absolute values.
- INC (incremental input): use this function to set incremental values for tool length, tool radius, tool wear compensation, and tool life.
- LEN OFST (tool length offset): the function is for automatically entering the tool length compensation values. Enter the absolute coordinate of each axis, and the corresponding tool length compensation values are automatically calculated (machine coordinates - input values). It avoids the risk of entering incorrect values and shortens the setting time.
- ABS OFST (absolute offset): writes the current absolute coordinates to the tool table.
- CLR ONE (clear single axis): clears all the offset data of one single axis.
- ABS/INC MODE (absolute / incremental mode): when you set Pr10059 to 1 (absolute) or 2 (incremental), the ABS/INC function key displays. When you set Pr10059 to 0, the function key is not available. You can enter the value and then press **ENTER** to determine whether this value is absolute or incremental.

The operation steps are as follows.

(1) Press ↑, ↓, ←, and → to move the cursor to the tool length field of a specific number.

(2) Enter absolute coordinates in the input field, press **LEN OFST**, and the controller automatically calculates the tool length compensation amount of the axis where the cursor is located.

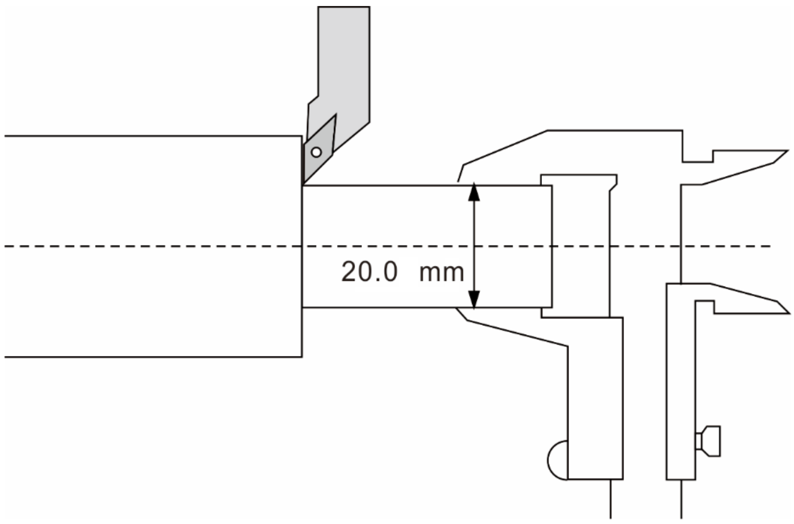
The calculation is: [Current machine coordinates] - [The input absolute coordinates] = [Tool length compensation amount]

The input absolute coordinates consist of signed values.

[Example of tool length compensation for X axis]

Enter 20.0 (the measured diameter of X axis after cutting in the unit of mm) to the input field of compensation value for X axis, press **LEN OFST**, and the system automatically calculates the tool length compensation value.

Important: when the cutting is complete, press **LEN OFST** without moving the X axis (do not change its machine coordinates).



8.3 Tool magazine management

8

This function is for managing the tool positions and their corresponding tool magazine numbers after tool change. When a different tool is used, the tool pocket positions and the corresponding tool numbers are recorded in the tool magazine data table.

You can view the recorded tool number corresponding to the tool pocket positions and also change the sequence of the tool number in the tool magazine data table. With parameter settings, you can enable the multi-magazine management function. The function of tool magazine management is only available in JOG mode, as shown in the following figure.

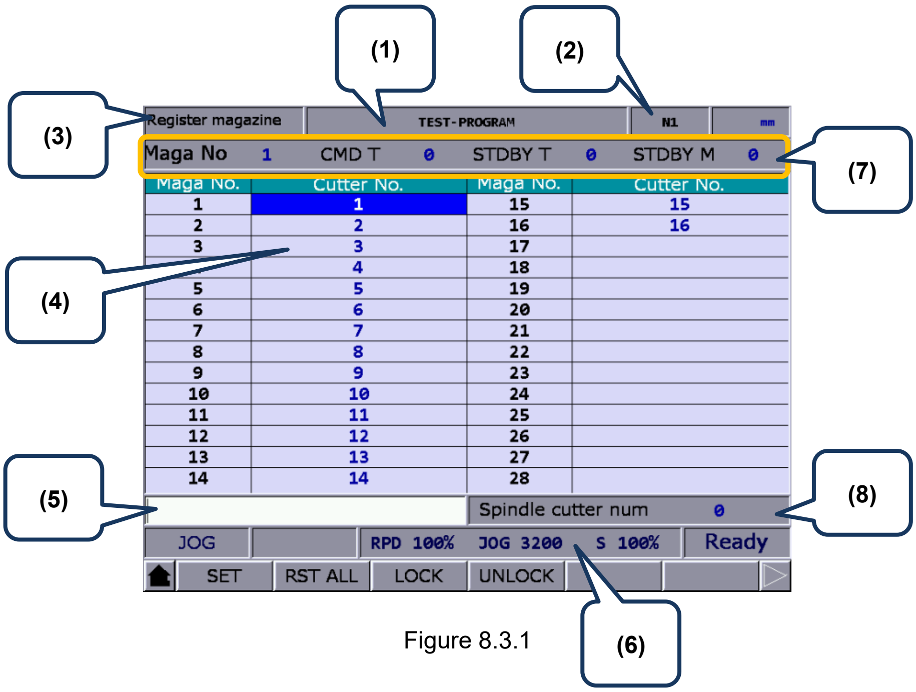


Figure 8.3.1

(1) Name of current program
(2) Currently executed program line
(3) Current group function
(4) Sequence of tool number
(5) Tool number input field
(6) Current override settings
(7) Number of tool magazine system
Tool number for current command
Current standby tool number
Current standby tool pocket number
(8) Spindle tool number

The operation steps for tool magazine setting are as follows.

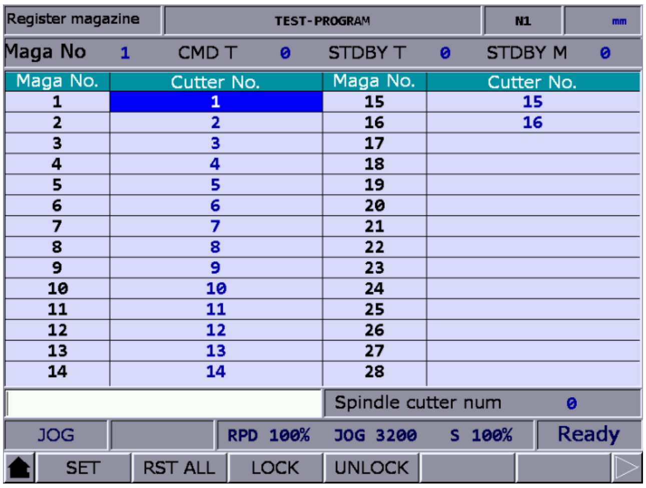
(1) Set the system to JOG mode.
(2) Press **OFS** to enter the OFS screen.
(3) Press **MAGA** to enter the tool magazine data setting screen.
(4) Press ⬆, ⬇, ⬅, and ➡ to move the cursor to a specific field.
(5) Enter the tool number and press **SET** (or **ENTER**) to change its corresponding tool magazine.

8

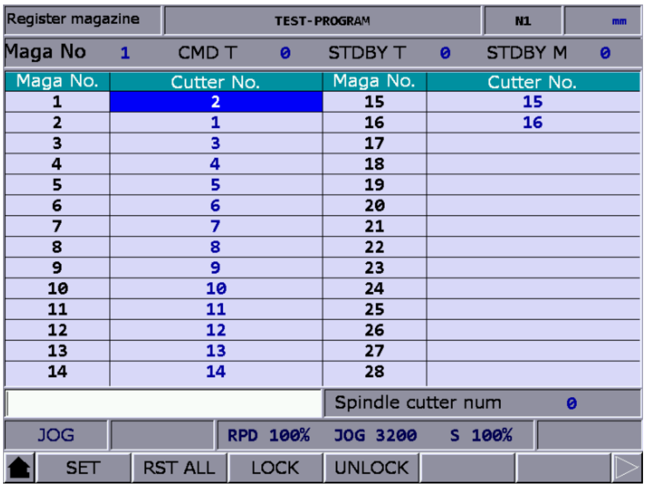
Example of changing tool number

When you specify a number that has already existed in the data table, the system automatically exchanges the one to be replaced with the one that is repeated. This is for ensuring that the tool numbers in each address of the tool magazine do not overlap, preventing the tool call error.

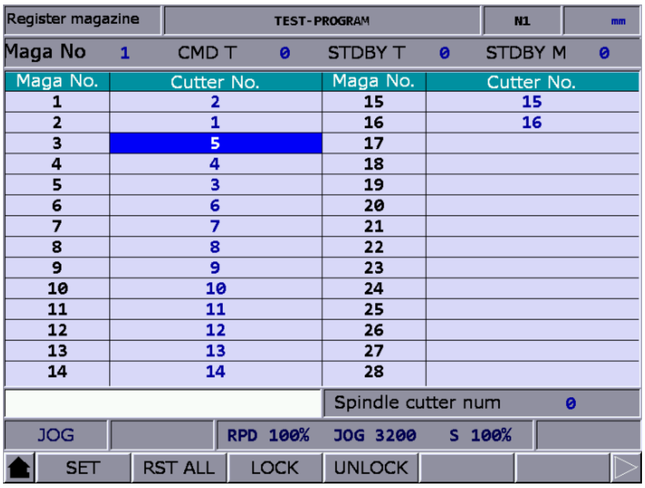
Workflow description:



(1) Initial status of the tool magazine. Tool numbers are arranged in sequence in accordance with the tool magazine numbers.



(2) If you set the tool number of tool magazine 1 to 2, then the tool number of tool magazine 2 becomes 1. That is, the tool numbers for tool magazines 1 and 2 are exchanged.



(3) If you set the tool number of tool magazine 3 to 5, then the tool number of tool magazine 5 becomes 3. That is, the tool numbers for tool magazines 3 and 5 are exchanged.

According to the above examples, the mechanism of tool number change can avoid the possibility of mistakenly calling the incorrect tool number.

- RST ALL (reset all): the tool magazine management provides the function of resetting the tool magazines by rearranging the tool numbers. After resetting, the records of changes in tool number are cleared. The tool numbers are arranged in sequence according to the tool magazine numbers. With this function, you can restore the data to default setting for troubleshooting tool number misplacement or tool number resetting. The operation steps are as follows.

  (1) Set the system to JOG mode.
  (2) Press **OFS** to enter the OFS screen.
  (3) Press **MAGA** to enter the tool magazine data setting screen.
  (4) Press **RST ALL** to reset the tool magazine data table.

- LOCK (tool magazine lock): use this function to lock the spare tool magazines. Tool numbers of the locked magazines cannot be called. If you use a command in the program to call a locked tool, the system enables the protection mechanism and displays an error message to stop the execution. This function is a preventive mechanism for checking the tool status during program execution, avoiding errors caused by incorrect tool call, such as damage to the latch of the tool magazine or interference to the magazine due to adjacent tools of large diameter. The data fields of the locked magazines are specified with different colors. The operation steps are as follows.

  (1) Set the system to JOG mode.
  (2) Press **OFS** to enter the OFS screen.
  (3) Press **MAGA** to enter the tool magazine data setting screen.
  (4) Press ↑, ↓, ←, and → to move the cursor to a specific data field.
  (5) Press **LOCK** to lock that magazine, as shown in Figure 8.3.2.

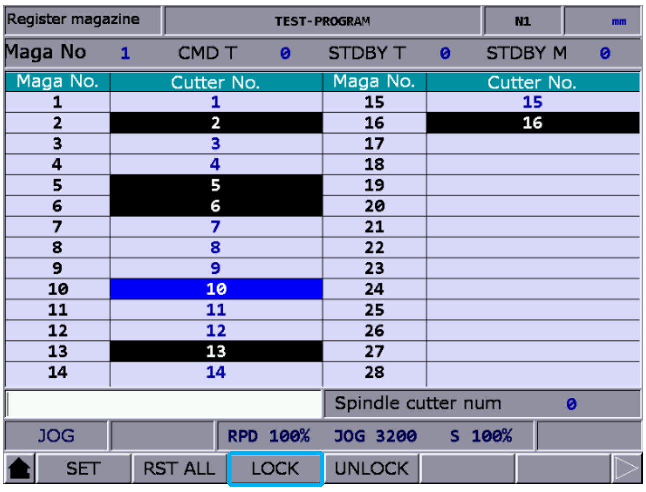


Figure 8.3.2

Example of locking the tool magazine

This function is used for preventing the tool of large diameter from damaging its adjacent tools by blocking the adjacent magazines to ensure the machine will not be damaged due to misoperation by personnel. By blocking the magazines adjacent to the magazine which carries a tool of large diameter, you can avoid executing inappropriate tool call program and thus prevent the possible collision caused by placing tools into the adjacent magazines.

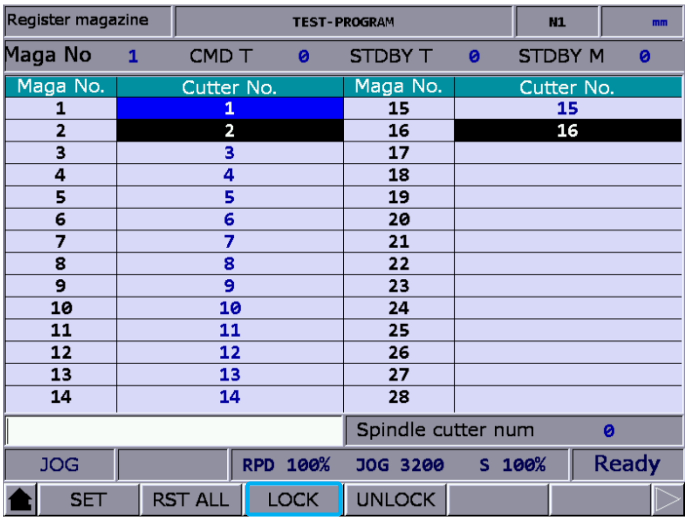


Figure 8.3.3

Assume that T1 is a tool of large diameter and the adjacent tools are T2 and T16, you can avoid its interference with the adjacent tools by locking T2 and T16, as shown in the above figure.

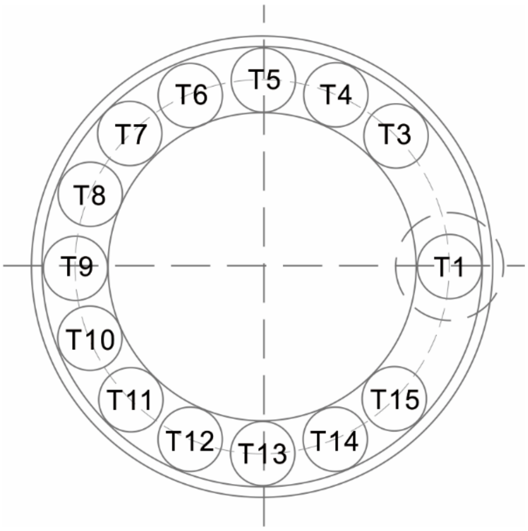


Figure 8.3.4

If T1 is a tool of large diameter, its interference with adjacent tools is as shown in Figure 8.3.4. With T2 and T16 locked, you cannot call tools in the two magazine positions.

- UNLOCK (tool magazine unlock): you can use this function to unlock the magazines. The operation steps are as follows.
  (1) Set the system to JOG mode.
  (2) Press **OFS** to enter the OFS screen.
  (3) Press **MAGA** to enter the tool magazine data setting screen.
  (4) Press ↑, ↓, ←, and → to move the cursor to the data field which has been locked.
  (5) Press **UNLOCK** to unlock the tool magazine. Or you can enter the same tool number to the data field of the locked magazine and press **ENTER** to unlock.

8

8.3.1 Multi-magazine management

In response to the application needs for managing multiple tool magazines, you can enable this function with tool magazine parameters after accessing the security authorization. You can specify the number of tool pockets for each tool magazine system according to the tool magazine specification as well as the tool numbers after resetting the tool magazine. The items [MAGA1] and [MAGA2] on the function bar are for dividing the management of the two different tool magazine systems. Contact the distributors for services of the multi-magazine management function.

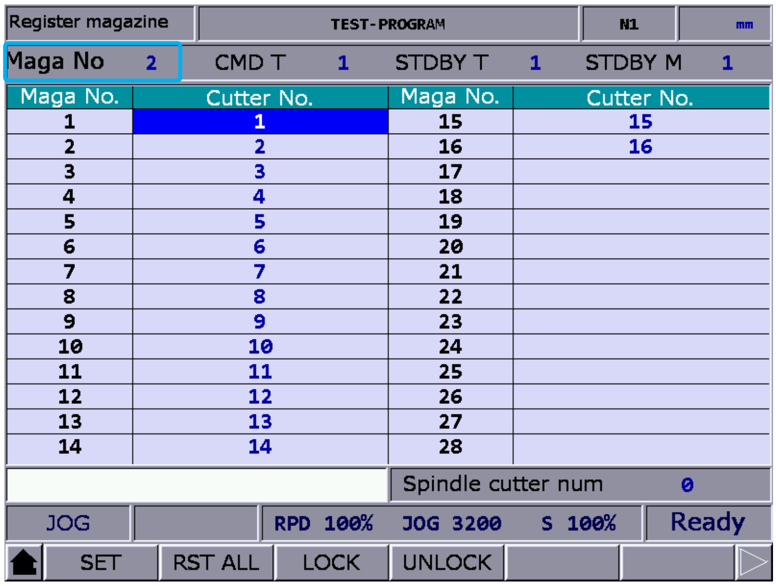


Figure 8.3.1.1

Note:

1. You can set the tool numbers only when the system is in JOG mode. Otherwise, the corresponding function bar is not displayed.
2. You have to access the security authorization in advance before setting or resetting the tool numbers.
3. Tool numbers in the same tool magazine system cannot be repeated. If you specify a number which already exists in the magazine, the system automatically changes the existing one with a non-repeating number. This is for ensuring the tool numbers in each address of the magazine do not overlap, preventing the tool call error.
4. The default spindle tool number is T0. Once tool T0 is placed into the tool magazine, its position in the magazine is recorded as T0 and cannot be locked. That is, when the displayed tool number is “0”, the LOCK function is disabled, and a dialog box appears and displays “T0 can’t be locked!”.

8.4 Macro variables

Using commands with variables, you can modify values, perform conditional operations, and input or output MLC data during program execution. There are four types of macro variables: local, global, non-volatile, and extension variables, with the data type as double word.

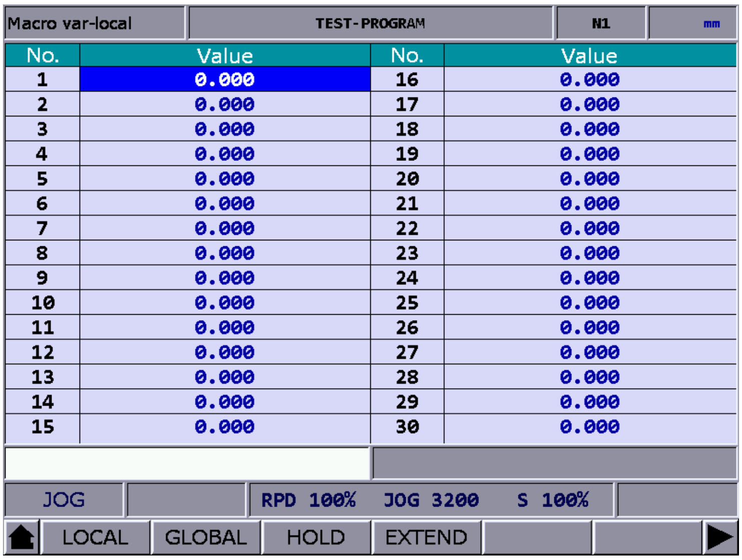


Figure 8.4.1

8.4.1 Local variables (LOCAL)

Local variables (#1 - #50) are available only in the current program.

The operation steps are as follows.

(1) Press **OFS** to enter the OFS screen.

(2) Press **MACRO** to display the variable entry screen.

(3) Press **LOCAL** and the screen is automatically switched to display the variable table starting with number 1.

(4) Press , , , and to move the cursor to a specific data field.

(5) Enter the value and press **ENTER** to complete the setting.

8.4.2 Global variables (GLOBAL)

Global variables (#51 - #250) are shared by main programs, subprograms, and macro programs. The operation steps are as follows.

(1) Press **OFS** to enter the OFS screen.
(2) Press **MACRO** to display the variable entry screen.
(3) Press **GLOBAL** and the screen is automatically switched to display the variable table starting with number 51.
(4) Press ↑, ↓, ←, and → to move the cursor to a specific data field.
(5) Enter the value and press **ENTER** to complete the setting.

8.4.3 Non-volatile variables (HOLD)

Non-volatile variables (#1601 - #1800) are for retaining the system status when power is off. The operation steps are as follows.

(1) Press **OFS** to enter the OFS screen.
(2) Press **MACRO** to display the variable entry screen.
(3) Press **HOLD** and the screen is automatically switched to display the variable table starting with number 1601.
(4) Press ↑, ↓, ←, and → to move the cursor to a specific data field.
(5) Enter the value and press **ENTER** to complete the setting.

8.4.4 Extension variables (EXTEND)

Up to 500 extension variables (#10001 - #10500) are available for the system. The operation steps are as follows.

(1) Press **OFS** to enter the OFS screen.
(2) Press **MACRO** to display the variable entry screen.
(3) Press **EXTEND** and the screen is automatically switched to display the variable table starting with number 10001.
(4) Press ↑, ↓, ←, and → to move the cursor to a specific data field.
(5) Enter the value and press **ENTER** to complete the setting.

Diagnosis (DGN) Group 9

The DGN group provides functions of machining information, user variables, system monitoring, and parameter importing / exporting, which help you to optimize the system.

The DGN group includes a variety of functions. Machining information, user variables, system monitoring, gain adjustment, and system information are for optimizing the system. MLC diagnosis is for monitoring the current status of the MLC devices in the system. Password setting allows you to assign security authorization for different system functions. In addition, system parameters can be imported and exported.

Note: bold function names in a box (such as **POS**) mean the keys on machine operation panel A;

bold function names (such as **CLR ALL**) mean the function keys of F1 - F6.

9.1 Machining information (PROCESS)

You can set the number of machined workpiece and number of workpiece to be machined, as well as clear the machining time and number of machined workpiece on the screen as shown in the following figure.

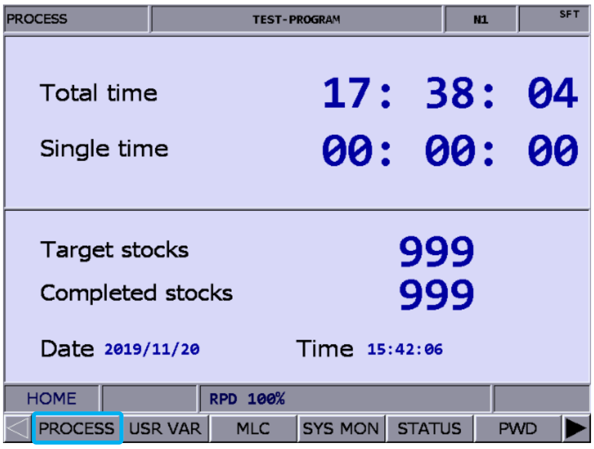


Figure 9.1.1

The operation steps are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **PROCESS** to enter the machining information screen.

(3) Press **SET NR** and a dialog box appears for you to enter the number of machining workpiece as shown in the following figure.

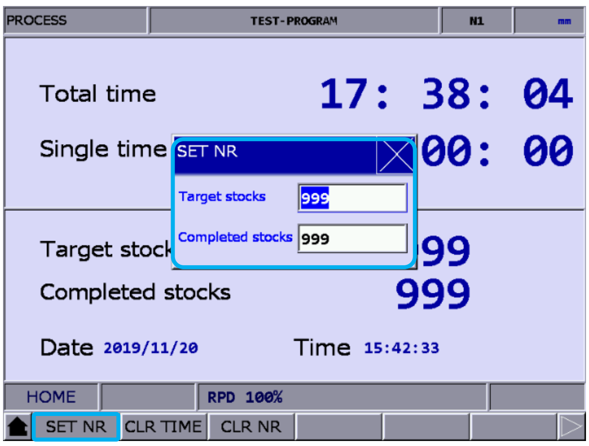


Figure 9.1.2

(4) Press ⬆ and ⬇ to move the cursor to a specific field.
(5) Enter a value within the range of 0 - 9999 and press **ENTER** to complete the setting.

In addition, you can clear the current machining time and number of machined workpiece on the machining information screen.

The operation steps for clearing the machining time are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PROCESS** to enter the machining information screen.
(3) Press **CLR TIME** and a dialog box appears for confirmation.
(4) Enter “Y” and press **ENTER** to clear the machining time for a single workpiece on the screen.

The operation steps for clearing the number of machined workpiece are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PROCESS** to enter the machining information screen.
(3) Press **CLR NR** and a dialog box appears for confirmation.
(4) Enter “Y” and press **ENTER** to clear the number of machined workpiece on the screen.

9

9.2 User variable (USR VAR)

The functions of user variable include system variable (SYS VAR), user variable (USR VAR), and machine variable (M VAR). You can use the function of system variable to monitor specific variables, and use the functions of user variable and machine variable to enter the names of registers (D512 - D1023) and display the corresponding data on the screen. With the displayed types of registers, you can easily control the corresponding devices by monitoring and changing the setting values of the registers (D512 - D1023).

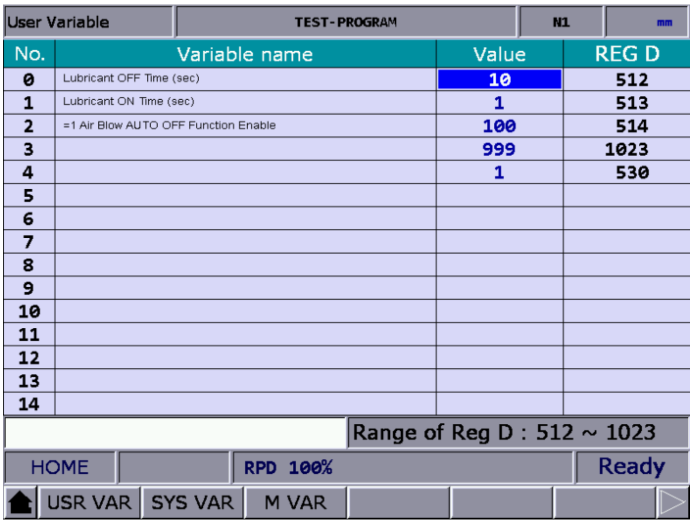


Figure 9.2.1

The operation steps for user variable and machine variable are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **USR VAR** to enter the variable monitoring screen.

(3) Press **USR VAR** or **M VAR** on the function bar in the second layer to enter the setting screen.

(4) Press ↑ and ↓ or **PAGE UP** and **PAGE DN** to move the cursor to a specific field.

(5) Enter the specified register number (D512 - D1023) and press **ENTER** to load the data in the register of the specified number.

(6) Move the cursor to the value field of the specified register, enter a valid value (range: 0 - 65535), and press **ENTER** to set the value.

(7) Press **US DEC**, **HEX**, **S DEC**, or **FLOAT** to select the data format display.

(8) To delete data, move the cursor to the data field and press **DEL** to delete the data.

9.3 MLC

This function displays the current status of each MLC device, so you can monitor and force On or Off each device. You can also check the system status, drive a certain MLC device, or edit the MLC. See Figure 9.3.1 for the MLC screen. MLC-related diagnostic functions include bit status, register status, device monitoring, MLC status switching, and MLC editing. The operation steps for these functions are described in the following sections.

9

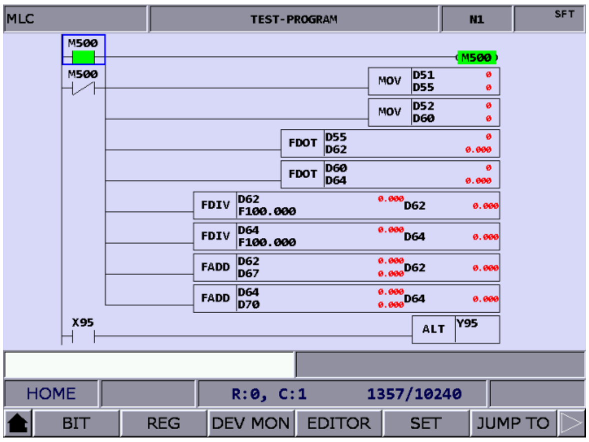


Figure 9.3.1

9.3.1 Bit (BIT)

MLC programs require a number of commands to trigger the devices to On or Off. Status of these devices is shown on the MLC Bit Device screen. This function is for displaying the bit type MLC devices, searching the device, and forcing the device to On or Off. The following operation steps take the M devices as an example.

(1) Press **DGN** to enter the DGN screen.

(2) Press **MLC** to display the function bar in the second layer.

(3) Press **BIT** to enter the bit device status display screen.

(4) Press **M** to switch to the status display for M devices as shown in the following figure. Move the cursor to a specific device field or search for the device by following Step (5).

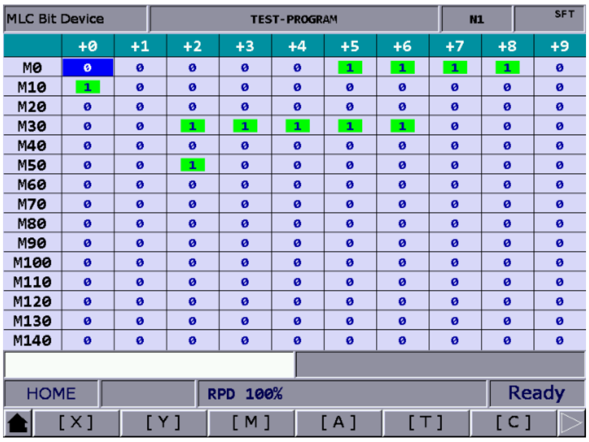


Figure 9.3.1.1

(5) Enter the device name, such as 107, and press **M** to search for the specified device, M107. To change the status of this device, set the system to non-AUTO mode. Refer to Step (6) for the operation steps.

(6) Specify the device which status is to be changed. Depending on its current status, enter "1" to force it to On or "0" to force it to Off, then press **ENTER**.

9.3.2 Register (REG)

Most of the CNC system functions are enabled by MLC programs. MLC devices are divided into bit type and word type. The following operation steps take the word type MLC device and T registers as an example.

(1) Press **DGN** to enter the DGN screen.

(2) Press **MLC** to display the function bar in the second layer.

(3) Press **REG** to enter the register device screen.

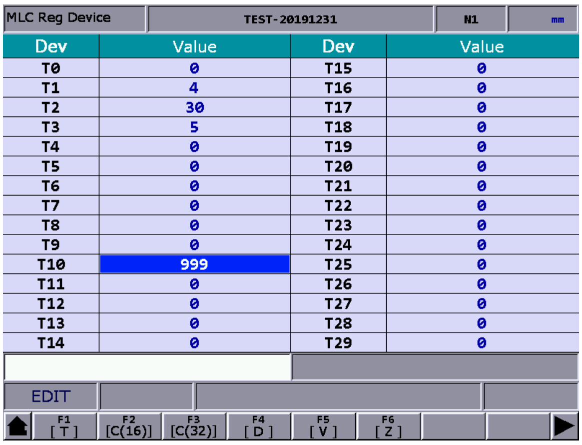


Figure 9.3.2.1

(4) Press **T** to enter the register T setting screen.

(5) Enter the device name, such as 10, and press **T** to search for the device, T10.

(6) Enter the value in the input field and press **ENTER** to complete the setting.

(7) Go to the last page of the function bar and press **US DEC**, **HEX**, **S DEC**, or **FLOAT** to select the data format display.

9.3.3 Device monitoring (DEV MON)

Up to 45 sets of device data can be monitored with this function. The operation steps are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **MLC** to display the function bar in the second layer.

(3) Press **DEV MON** to display the corresponding screen as shown in Figure 9.3.3.1.

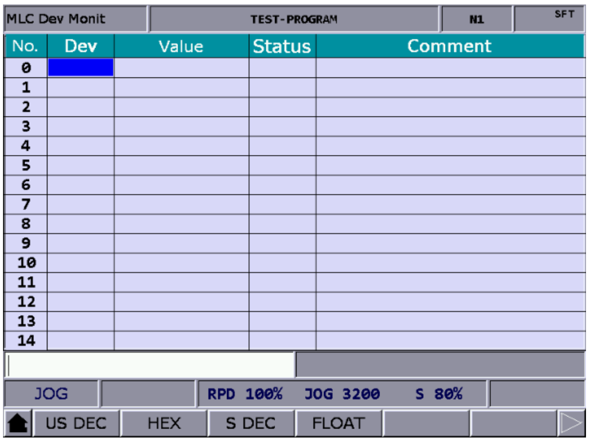


Figure 9.3.3.1

(4) Enter the name of the device to be monitored as shown in Figure 9.3.3.2. You can enter up to 45 device names.

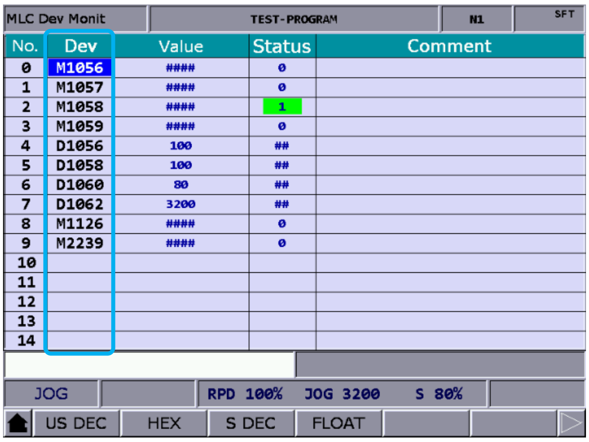


Figure 9.3.3.2

Dev (device): when the cursor is located in this field, you can enter the name of the device to be monitored.

Value: move the cursor to this field to set the data of the device.

Status: enter “0” or “1” to set the device status.

In addition, you can switch the data format display according to the requirements by using the functions of **US DEC**, **HEX**, **S DEC**, and **FLOAT**. The data in Figure 9.3.3.3 are in hexadecimal format and the data in Figure 9.3.3.4 are in floating format.

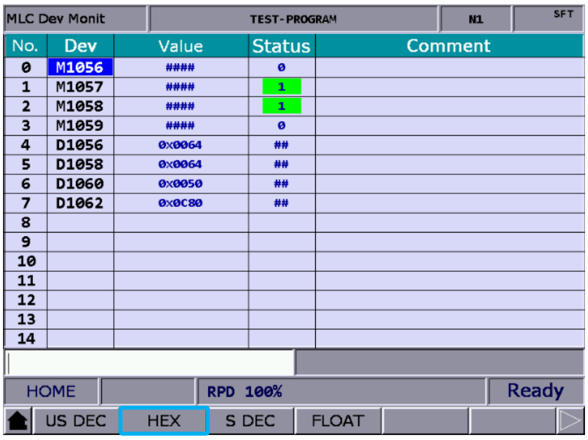


Figure 9.3.3.3

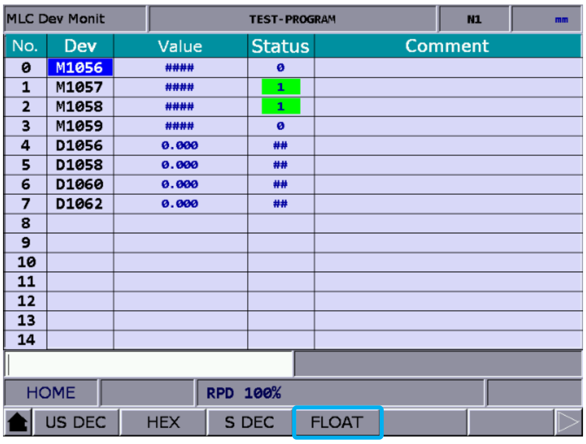


Figure 9.3.3.4

9.3.4 Line search (JUMP TO)

Use this function to search for a specific line according to the entered line number of the MLC program.

(1) Press $\boxed{\textbf{DGN}}$ to enter the DGN screen.

(2) Press **MLC** to display the function bar in the second layer.

(3) Enter a specific line number of the MLC program and press **JUMP TO** to go to the target line.

9.3.5 Editor (EDITOR)

Use this function to manage and edit the MLC programs. You can directly edit the MLC programs on the controller interface with the system set to EDIT mode.

- Basic MLC commands

  Basic MLC commands, including LD, LDI, LDP, LDF, OUT, APP, —, and ｜, are created with the function of MLC editing, as shown in Figure 9.3.5.1.

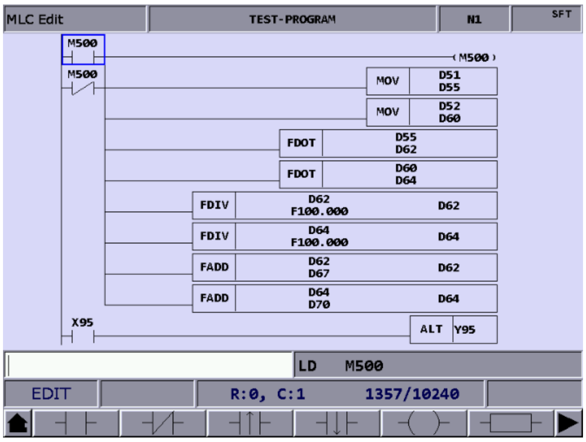


Figure 9.3.5.1

The operation steps for creating command LD are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **MLC** to display the function bar in the second layer.

(3) Press **EDITOR** to enter the MLC Edit screen as shown in Figure 9.3.5.1.

(4) Press ↑, ↓, ←, and → to move the cursor to the field to be edited.

(5) Enter the device name and press **LD** to complete creating the command.

The above steps are also applicable for creating LDI, LDP, LDF, OUT, APP, — , and ｜ commands.

To specify the values for the basic commands, press **TABLE** to display the MLC table as shown in the following figure.

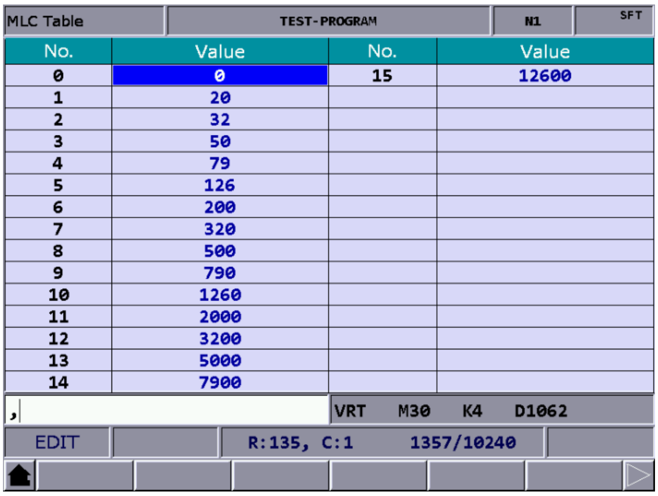


Figure 9.3.5.2

- Editing (CUT, COPY, PASTE)

These editing functions are MLC-specific, with which you can delete, cut, or copy a single line of program. Or you can use the SELECT function to delete, cut, or copy a certain section of the MLC program. After completing the editing, use the SAVE function to recompile and save the edited MLC program. The operation steps for editing MLC programs are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **MLC** to display the function bar in the second layer.

(3) Press **EDITOR** to enter the MLC Edit screen as shown in Figure 9.3.5.1.

(4) Press ↑, ↓, ←, and → to move the cursor to the field to be edited.

(5) Repeatedly press ▶ to display the function bar on the last page in this layer.

(6) Press the corresponding function key such as **CUT** to edit. When editing MLC programs, press the corresponding function keys according to the editing requirements, including **SELECT**, **DEL**, **CUT**, **COPY**, **PASTE**, **ADD LN**, and **DEL LN**.

- Symbol

  Use this function to search, delete, copy, and paste the MLC devices. MLC devices are represented with the symbols X, Y, M, A, T, C, D, P, and I. The operation steps are as follows.

  (1) Press $\boxed{\textbf{DGN}}$ to enter the DGN screen.

  (2) Press **MLC** to display the function bar in the second layer.

  (3) Press **EDITOR** to enter the MLC Edit screen as shown in Figure 9.3.5.1.

  (4) Press ⬆, ⬇, ⬅, and ➡ to move the cursor to the field to be edited.

  (5) Press ▶ to display the function bar on the third page.

  (6) Press **SYMBOL** to display the corresponding function bar.

  (7) Press the function key, such as **X**, to display the list of corresponding devices and use the functions of delete, copy, or paste as required.

  Note: the above steps are applicable to other device symbols.

- Save, import, and export MLC

  After editing the MLC program, use the SAVE function to recompile and save the program. Then restart the system to update the MLC program. In addition, use the corresponding function keys (IMPORT / EXPORT) to import or export MLC files.

9.3.6 Operation (SET)

The system runs the MLC program right after starting. To manually switch the execution status, use this function to stop the MLC program. This function is for switching the MLC program status to On or Off, which is usually used for testing or checking the MLC devices in the system. The operation steps are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **MLC** to display the function bar in the second layer.

(3) Press **SET** to switch to the screen of MLC execution status.

(4) Press **RUN** / **STP** to force switch the MLC program status.

Note: after stopping the MLC program, you can see the status of “MLC Stop” in the system status field.

In addition, you can force the MLC device to On or Off using the corresponding functions.

- The operation steps for forcing the device status to On are as follows.
  - (1) Press **DGN** to enter the DGN screen.
  - (2) Press **MLC** to display the function bar in the second layer.
  - (3) Press **SET** to switch to the screen of MLC execution status.
  - (4) Press ↑, ↓, ←, and → to move the cursor to a specific device.
  - (5) Press **ON** to switch the MLC status to On.

- The operation steps for forcing the device status to Off are as follows.
  - (1) Press **DGN** to enter the DGN screen.
  - (2) Press **MLC** to display the function bar in the second layer.
  - (3) Press **SET** to switch to the screen of MLC execution status.
  - (4) Press ↑, ↓, ←, and → to move the cursor to a specific device.
  - (5) Press **OFF** to switch the MLC status to Off.

9.4 System monitoring (SYS MON)

This function categorizes the various calculation results of the system and displays them according to their types for your reference.

9

9.4.1 Servo monitoring (SRV MON)

This function displays the servo drive status on the screen of the system, from which you can check the information about the channel port number and servo status of each axis. As shown in Figure 9.4.1.1, both the spindle and Z axis are in the Off status and both X and Y axes are in the On status.

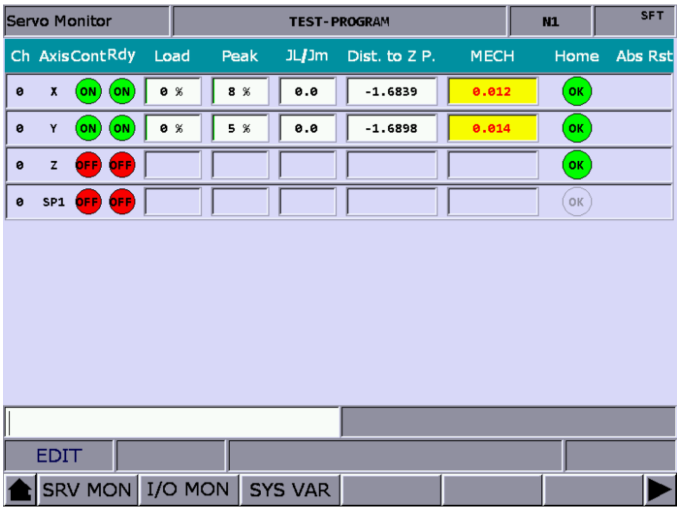


Figure 9.4.1.1

The operation steps are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **SYS MON** to switch to the system monitoring screen.

(3) Press **SRV MON** to display the servo monitoring screen.

9.4.2 I/O monitoring (I/O MON)

The NC series system can add the control switches for external devices with the I/O extension modules. You can monitor the status of the I/O extension control board connected to the system on the I/O monitoring screen. The operation steps are as follows.

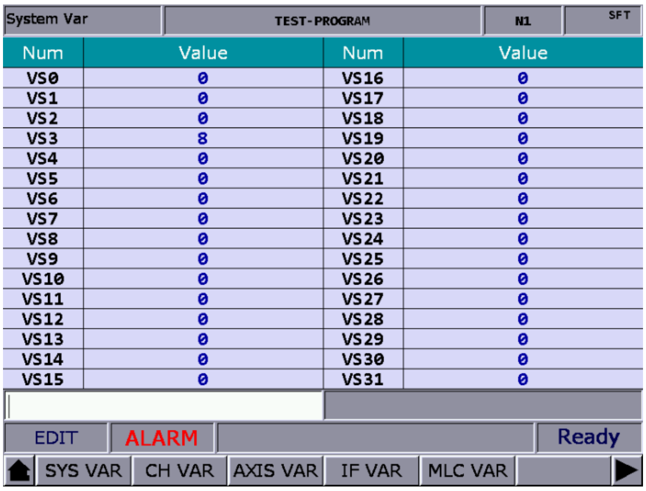
(1) Press **DGN** to enter the DGN screen.

(2) Press **SYS MON** to switch to the system monitoring screen.

(3) Press **I/O MON** to display the I/O extension module status monitoring screen.

9.4.3 Variable monitoring (VAR MON)

- System variables (SYS VAR): VS0 - VS31 and VS100 - VS131. The operation steps are as follows.
  (1) Press **DGN** to enter the DGN screen.
  (2) Press **SYS MON** to switch to the system monitoring screen.
  (3) Press ▶ to display the function bar on the second page.
  (4) Press **VAR MON** to display the variable monitoring screen.
  (5) Press **SYS VAR** to display the system variable screen.
  (6) Press **PAGE UP** and **PAGE DN** to scroll to the page with the specified variable.
  (7) You can also enter the full name of a specific system variable and press **ENTER**, or enter the variable number and press **SYS VAR** to search for and display the specified system variable.



Num	Value	Num	Value
VS0	0	VS16	0
VS1	0	VS17	0
VS2	0	VS18	0
VS3	8	VS19	0
VS4	0	VS20	0
VS5	0	VS21	0
VS6	0	VS22	0
VS7	0	VS23	0
VS8	0	VS24	0
VS9	0	VS25	0
VS10	0	VS26	0
VS11	0	VS27	0
VS12	0	VS28	0
VS13	0	VS29	0
VS14	0	VS30	0
VS15	0	VS31	0

Figure 9.4.3.1

- Channel variables (CH VAR): VC0 - VC31, VC100 - VC131, and VC200 - VC231. The operation steps are as follows.
  (1) Press **DGN** to enter the DGN screen.
  (2) Press **SYS MON** to switch to the system monitoring screen.
  (3) Press ▶ to display the function bar on the second page.
  (4) Press **VAR MON** to display the variable monitoring screen.
  (5) Press **CH VAR** to display the channel variable screen.
  (6) Press **PAGE UP** and **PAGE DN** to scroll to the page with the specified variable.
  (7) You can also enter the full name of a specific channel variable and press **ENTER**, or enter the variable number and press **CH VAR** to search for and display the specified channel variable.

- Axis variables (AXIS VAR): VA0 - VA31, VA100 - VA131, and VA200 - VA231.

  The operation steps are as follows.

  (1) Press **DGN** to enter the DGN screen.

  (2) Press **SYS MON** to switch to the system monitoring screen.

  (3) Press ▶ to display the function bar on the second page.

  (4) Press **VAR MON** to display the variable monitoring screen.

  (5) Press **AXIS VAR** to display the axis variable screen.

  (6) Press **PAGE UP** and **PAGE DN** to scroll to the page with the specified variable.

  (7) You can also enter the full name of a specific axis variable and press **ENTER**, or enter the variable number and press **AXIS VAR** to search for and display the specified axis variable.

- Interface variables (IF VAR): VH0 - VH31, VH200 - VH231, VH400 - VH431, and VH800 - VH863. The operation steps are as follows.

  (1) Press **DGN** to enter the DGN screen.

  (2) Press **SYS MON** to switch to the system monitoring screen.

  (3) Press ▶ to display the function bar on the second page.

  (4) Press **VAR MON** to display the variable monitoring screen.

  (5) Press **IF VAR** to display the interface variable screen.

  (6) Press **PAGE UP** and **PAGE DN** to scroll to the page with the specified variable.

  (7) You can also enter the full name of a specific interface variable and press **ENTER**, or enter the variable number and press **IF VAR** to search for and display the specified interface variable.

- MLC variables (MLC VAR): VM0 - VM49. The operation steps are as follows.

  (1) Press **DGN** to enter the DGN screen.

  (2) Press **SYS MON** to switch to the system monitoring screen.

  (3) Press ▶ to display the function bar on the second page.

  (4) Press **VAR MON** to display the variable monitoring screen.

  (5) Press **MLC VAR** to display the MLC variable screen.

  (6) Press **PAGE UP** and **PAGE DN** to scroll to the page with the specified variable.

  (7) You can also enter the full name of a specific MLC variable and press **ENTER**, or enter the variable number and press **MLC VAR** to search for and display the specified MLC variable.

9

9.5 Password setting (PWD)

In order to effectively control the operation security of the system functions, you can use this function to assign different levels of authorization for the system (system maintenance), machine (mechanical devices), and user (operation). This prevents unauthorized users from changing the system settings and thus affecting the system operation.

9.5.1 System security (S SCP)

This function includes security lock (LOCK), security unlock (UNLOCK), and system check (SYS CHECK). The password must be four characters containing at least one letter and one number (no special characters). The operation steps for locking and unlocking the system security are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) Press **S SCP** to display the corresponding function bar.
(4) When the system security is in the unlocked status, press **LOCK** to immediately lock all system-related functions.
(5) When the system security is in the locked status, press **UNLOCK**, and a dialog box appears for you to enter the valid password.
(6) After entering the password, press **ENTER** to unlock the system.

The operation steps for system check are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) Press **S SCP** to display the corresponding function bar.
(4) When the system is in the unlocked status, press **SYS CHECK** to check the status of the items. If the check box for an item is selected, it means that item is in error.

9.5.2 Machine security (M SCP)

This function includes password change (PWG CHG), security lock / security unlock (LOCK / UNLOCK), and user reset (RST U1 and RST U2). The password must be four characters containing at least one letter and one number (no special characters). The operation steps for changing the machine security are as follows.

9

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) Press **M SCP** to switch to the function bar of machine security.
(4) Press **PWD CHG** and a dialog box appears as shown in Figure 9.5.2.1. Enter the old password, new password, and retype the new password for confirmation.
(5) Press **ENTER** to complete changing the password.

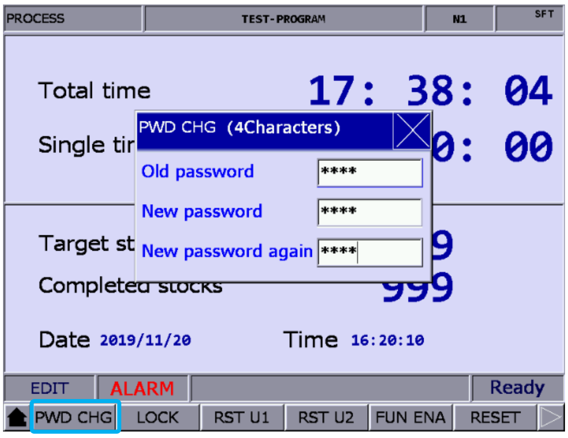


Figure 9.5.2.1

The operation steps for unlocking the machine security are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) Press **M SCP** to switch to the function bar of machine security.
(4) When the machine security is in the locked status, press **UNLOCK** and a dialog box appears for you to enter the valid password.
(5) Enter the valid password and press **ENTER** to unlock the machine security.

Note: the default password for machine security is 0000 which is not assigned to any authorization level, meaning that users of any level can access the machine-related functions. You need to change the default password of 0000 to enable the machine security, and only users with the authorization can access the machine-related functions.

The operation steps for locking the machine security are as follows.

(1) Press DGN to enter the DGN screen.

(2) Press **PWD** to switch to the function bar of password setting.

(3) Press **M SCP** to switch to the function bar of machine security.

(4) When the machine security is in the unlocked status, press **LOCK** to immediately lock all machine-related functions.

The function of user reset allows the equipment supplier to reset the user's password. If you forget the password, ask the equipment supplier to reset the password to the default. This function is only available when the password is changed from the default. The operation steps are as follows.

(1) Press DGN to enter the DGN screen.

(2) Press **PWD** to switch to the function bar of password setting.

(3) Press **M SCP** to switch to the function bar of machine security.

(4) Press **RST U1** or **RST U2** to reset the user's password.

Function enabling (FUN ENA) allows the equipment supplier to enable or disable the group functions. Once the selected group function check box is cleared, the group function is disabled after restarting. The operation steps are as follows.

(1) Press DGN to enter the DGN screen.

(2) Press **PWD** to switch to the function bar of password setting.

(3) Press **M SCP** to switch to the function bar of machine security.

(4) Press **FUN ENA** to enter the setting screen for enabling or disabling the group functions.

(5) Press ↑ and ↓ to move the cursor to the check box of the group function to be cleared and press ENTER to clear the check box. Then press **OK** and restart the system for the changes to take effect.

(6) Continued from Step (5), to maintain the original settings, press **CANCEL** to exit the screen and discard the previous settings.

(7) To restore to the system default setting, press **DEFAULT**.

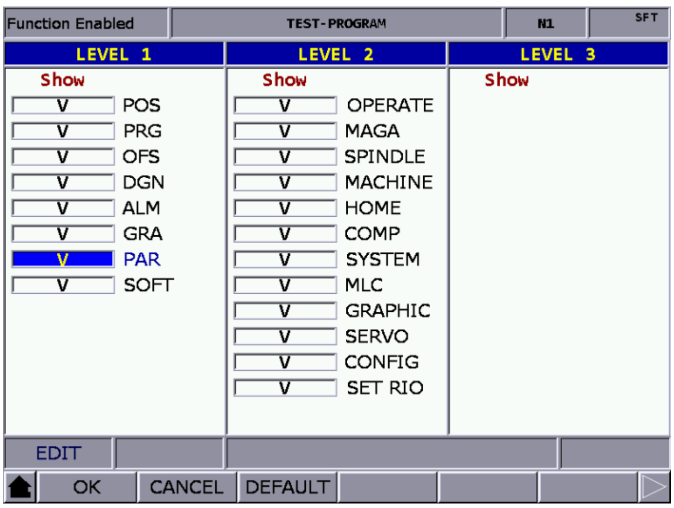


Figure 9.5.2.2

The function of restoring to default (DEFAULT) allows users to restore the system with the system backup file when the system is in error or the system data is seriously damaged. In the Default screen, if the check box is selected, it means the data of that item is damaged. You can use this function to restore the data of that item. This function is available only when you are authorized. The operation steps are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) Press **M SCP** to switch to the function bar of machine security.
(4) Press **DEFAULT** to enter the corresponding screen and press ↑, ↓, ←, and → to move the cursor. Press **ENTER** to select the item to be restored.
(5) Clear the check box: move the cursor to the selected item and press **ENTER** to clear the check box.
(6) Press **OK** to restore the system.

9.5.3 User security (U1 SCP & U2 SCP)

This function includes **U1 SCP** and **U2 SCP**. The functions of user security include password change (PWD CHG), security lock (LOCK), and security unlock (UNLOCK). The password must be four characters containing at least one letter and one number (no special characters). Take U1 SCP for example, the operation steps for changing the user password are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) When U1 SCP is locked, press **U1 SCP** and a dialog box appears for you to enter the password for U1 SCP.
(4) Enter the valid password for U1 SCP, press **ENTER**, and U1 SCP is unlocked and the corresponding function bar is displayed.
(5) Press **PWD CHG** and a dialog box appears. Enter the old password, new password, and retype the new password for confirmation.
(6) Press **ENTER** to complete changing the password.

The operation steps for unlocking the user security are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) When U1 SCP is locked, press **U1 SCP** and a dialog box appears for you to enter the password for U1 SCP.
(4) Enter the valid password for U1 SCP, press **ENTER**, and U1 SCP is unlocked and the corresponding function bar is displayed.

The operation steps for locking the user security are as follows.

(1) Press DGN to enter the DGN screen.

(2) Press **PWD** to switch to the function bar of password setting.

(3) When U1 SCP is unlocked, press **U1 SCP** and a dialog box appears for you to enter the password for U1 SCP.

(4) Press **LOCK** to lock U1 SCP.

Note: the function of user security is the same as that of machine security. The default password for user security is 0000 which is not assigned to any authorization level. You need to change the default password of 0000 to enable the user security.

9.5.4 Time limit (EXPIRE)

For specific situations that require a time limit on usage, you can set the operation time for the controller by security authorization. After the time limit is set, the available duration (hours / days) is automatically controlled by the system. When this function is enabled, you can only remove or reset the time limit with the proper authorization. When the time limit is not set or is disabled, no expiration date is displayed in the Deadline field on the screen, as shown in Figure 9.5.4.1. Once you set the time limit and do not disable it, the expiration date is displayed in the Deadline field, as shown in Figure 9.5.4.2.

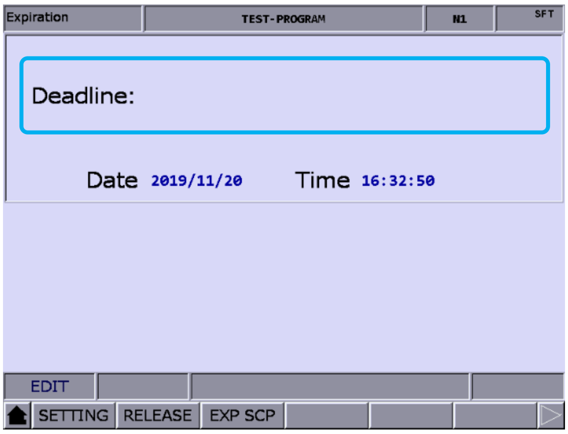


Figure 9.5.4.1

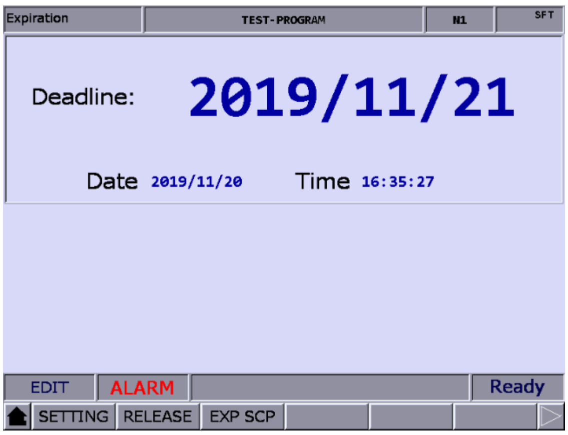


Figure 9.5.4.2

You can see the controller's operation expiration date and time on this screen. When the duration is up, the system will be locked, meaning that any execution of G-code program (in both AUTO and MDI modes) is prohibited. The execution will not resume until the time limit setting is disabled or extended. If the time limit expires, contact the distributor or equipment supplier for entering the valid password to remove the time limit.

You can set the time limit only when the function is not enabled. The operation steps for setting the time limit are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) Press **EXPIRE** to display the expiration information.
(4) Press **SETTING** to display the setting screen of time limit.
(5) Enter a valid password to set the time limit for the controller's operating duration.

The operation steps for removing the time limit are as follows (contact the distributor or equipment supplier for services).

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) Press **EXPIRE** to display the expiration information.
(4) Press **RELEASE** and a dialog box appears for you to enter the activation code as shown in Figure 9.5.4.3.

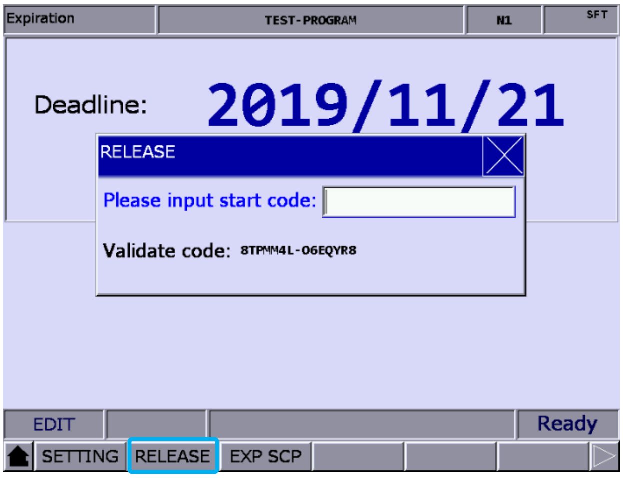


Figure 9.5.4.3

(5) Press **ENTER** and cycle power to the system to remove the time limit.

Note: after the time limit is removed, no expiration date is displayed in the Deadline field as shown in Figure 9.5.4.1, which means no time limit is set for the system.

The proper authorization is required to lock or unlock the EXP SCP function. When the function is locked, enter the valid security password to unlock it. When the function is unlocked, you can use all the functions for the time limit setting. The functions include password change (PWD CHG), security lock (LOCK), and security unlock (UNLOCK). The password must be four characters containing at least one letter and one number (no special characters).

The operation steps for changing the password for EXP SCP are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) Press **EXPIRE** to display the expiration information.
(4) When EXP SCP is locked, press **EXP SCP** and a dialog box appears for you to enter the password for EXP SCP.
(5) Enter the valid password, press **ENTER**, and EXP SCP is unlocked and the corresponding function bar appears.
(6) Press **PWD CHG** and a dialog box appears. Enter the old password, new password, and retype the new password for confirmation.
(7) Press **ENTER** to complete changing the password.

The operation steps for unlocking the expiration security are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) Press **EXPIRE** to display the expiration information.
(4) When EXP SCP is locked, press **EXP SCP** and a dialog box appears for you to enter the password for EXP SCP.
(5) Enter the valid password, press **ENTER**, and EXP SCP is unlocked and the corresponding function bar appears.

The operation steps for locking the expiration security are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press **PWD** to switch to the function bar of password setting.
(3) Press **EXPIRE** to display the expiration information.
(4) When EXP SCP is unlocked, press **EXP SCP** to display the corresponding function bar.
(5) Press **LOCK** to lock EXP SCP.

9.6 System information (STATUS)

This function provides information about the firmware and hardware versions of the system. You can maintain and optimize the system according to the version information displayed on the screen. The functions include system status, firmware serial number, hardware serial number, and equipment information.

The operation steps for system status display are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **STATUS** to enter the system information screen.

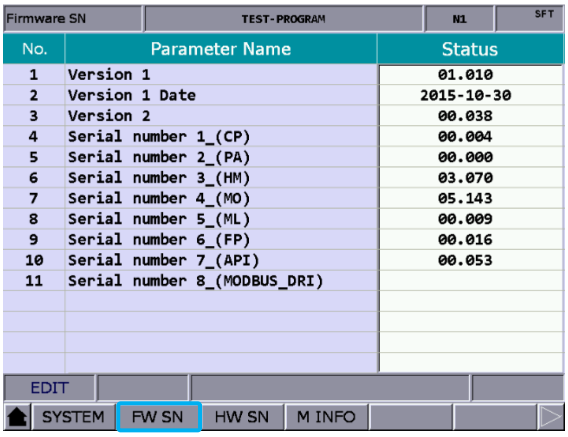
(3) Press **SYSTEM** to display the system status screen.

The firmware serial number display includes the firmware version of the system. The operation steps for displaying the firmware serial number are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **STATUS** to enter the system information screen.

(3) Press **FW SN** to display the firmware versions of the system as shown in the following figure.



No.	Parameter Name	Status
1	Version 1	01.010
2	Version 1 Date	2015-10-30
3	Version 2	00.038
4	Serial number 1_(CP)	00.004
5	Serial number 2_(PA)	00.000
6	Serial number 3_(HM)	03.070
7	Serial number 4_(MO)	05.143
8	Serial number 5_(ML)	00.009
9	Serial number 6_(FP)	00.016
10	Serial number 7_(API)	00.053
11	Serial number 8_(MODBUS_DRI)	

Figure 9.6.1

The operation steps for displaying the hardware serial number are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **STATUS** to enter the system information screen.

(3) Press **HW SN** to display the hardware version.

The operation steps for displaying the device information are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press **STATUS** to enter the system information screen.

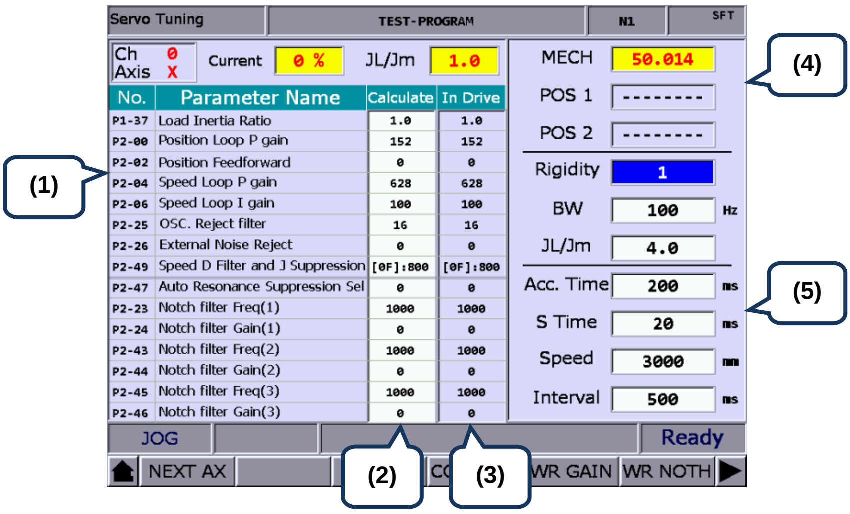
(3) Press **M INFO** to display the device information screen.

(4) You can enter the device information on the screen or press **DEL** to delete the information in the field where the cursor is located.

9

9.7 Gain tuning (TUNING)

The auto tuning function enables the NC system and servo drive to provide optimized motion control for meeting different machine characteristics. The NC series controller reads the initial servo parameters and calibrates the motion control with the gain tuning function. The results of gain tuning are returned to the servo drive, so the control parameters of the controller and drive are consistent. This facilitates the gain tuning procedure and maintains high-precision control of the system. The Servo Tuning screen is as shown in the following figure and the operation steps for each subordinate function are as follows.



(1) Servo parameter number: number and name of servo parameters

(2) Results after tuning: displays the calculation results of auto tuning

(3) System settings: displays the current servo settings

(4) Position setting: Position 1 / Position 2

(5) Tuning conditions

- Next axis (NEXT AX): for switching to another axis for gain tuning setting. You need to perform auto tuning for each axis separately, so after completing the setting of one axis, use this function to switch to another axis and continue auto tuning. The operation steps are as follows.
  (1) Press **DGN** to enter the DGN screen.
  (2) Press ▶ to display the function bar on the next page.
  (3) Press **TUNING** to enter the auto tuning setting screen.
  (4) If you need to set the gain parameters for other axes, press **NEXT AX** to switch to the specified axis.

- Read servo parameters (READ): accesses the parameter values from the servo and writes them in the Calculate fields. The operation steps are as follows.
  (1) Press **DGN** to enter the DGN screen.
  (2) Press ▶ to display the function bar on the next page.
  (3) Press **TUNING** to enter the auto tuning setting screen.
  (4) Press **READ** to read the servo parameters.

- RUN, JOG←, JOG→, POS1, POS2: these functions are for setting the operation of auto tuning. Use these functions to enable auto tuning, and set and operate the positioning direction. The operation steps for the continuous operation of a single axis are as follows.
  (1) Press **DGN** to enter the DGN screen.
  (2) Press ▶ to display the function bar on the next page.
  (3) Press **TUNING** to enter the auto tuning setting screen.
  (4) Press ▶ to display the function bar on the next page.
  (5) Press **JOG←** to move to the left positioning point.
  (6) Press **POS1** to set the left positioning point.
  (7) Press **JOG→** to move to the right positioning point.
  (8) Press **POS2** to set the right positioning point. The movement is now limited to between Position 1 and Position 2.
  (9) Press **RUN** to start auto tuning.
  (10) During auto tuning, press **STOP** to compete tuning. The system automatically calculates the best gain value.

- Gain calculation (COMPUTE): to change the parameter values of rigidity, bandwidth, or inertia to accommodate the machine characteristics, you can use this function to calculate and generate the results of gain tuning. The operation steps for calculating the gain value of a single axis are as follows.
  (1) Press **DGN** to enter the DGN screen.
  (2) Press ▶ to display the function bar on the next page.
  (3) Press **TUNING** to enter the auto tuning setting screen.

(4) Press ⬆ and ⬇ to move the cursor to the fields of Rigidity, BW, and JL/Jm to set the parameters.

(5) Press **COMPUTE** to calculate the tuning results.

- Gain value writing (WR GAIN), resonance value writing (WR NOTH): the system automatically calculates the gain values after auto tuning is finished and the motion stops. If the values after auto tuning meet the expectation, you can use these functions to write the new parameter values to the servo. The operation steps are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press ▶ to display the function bar on the next page.

(3) Press **TUNING** to enter the auto tuning setting screen.

(4) After tuning, the results are automatically calculated.

(5) Press **WR GAIN** to write the corresponding gain parameters to the servo.
Press **WR NOTH** to write the new parameter values for resonance suppression to the servo.

Note:

1. You have to write the results of auto tuning to the servo for the values to take effect.
2. After writing the gain values and resonance values, the servo parameters are updated and the previous settings cannot be restored. Thus, double check before writing the values.

- Calibration for tapping (TAP RIV): in tapping applications, calibrate the machine and servo with this function.

If you use Delta servo products for the spindle, the operation steps for TAP SET(1) are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press ▶ to display the function bar on the next page.

(3) Press **TUNING** to enter the auto tuning setting screen.

(4) Complete the gain tuning for the servo axes X, Y, and Z, and the spindle.

(5) Press ▶ to switch to the function bar on the last page.

(6) Press **TAP RIV** to switch to the corresponding screen.

(7) Press **TAP SET(1)** and a confirmation window appears. Enter “Y” and press **ENTER** to complete calibrating the machine for tapping applications.

If you use Delta AC inverter or servo products of other brands for the spindle, the operation steps for TAP SET(2) are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press ▶ to display the function bar on the next page.

(3) Press **TUNING** to enter the auto tuning setting screen.

(4) Complete the gain tuning for the servo axes X, Y, and Z, and the spindle.

(5) Press ▶ to switch to the function bar on the last page.

(6) Press **TAP RIV** to switch to the corresponding screen.

(7) Press **TAP SET(2)** and a confirmation window appears. Enter “Y” and press **ENTER**, then a dialog box appears for you to enter the gain value for spindle control. Enter an appropriate value and press **ENTER**, then the system adjusts the tapping setting for the machine according to the set value.

- Servo parameters (SERVO): for displaying and setting servo parameters in the Servo Tuning screen. The operation steps are as follows.
  (1) Press **DGN** to enter the DGN screen.
  (2) Press ▶ to display the function bar on the next page.
  (3) Press **TUNING** to enter the auto tuning setting screen.
  (4) Repeatedly press ▶ to switch to the function bar on the last page.
  (5) Press **SRV MON** to display the servo parameter screen.
  (6) Move the cursor to the field of the parameter to be edited, enter the value, and press **ENTER** to complete the setting.

9.8 Import (IMPORT)

With the proper authorization, you can use this function to import the backup parameters to the NC system. The operation steps for importing parameters are as follows.

(1) Press **DGN** to enter the DGN screen.
(2) Press ▶ to display the function bar on the next page.
(3) Press **IMPORT** to display the file manager (FILE) window as shown in Figure 9.8.1. Press ⬆ and ⬇, select the directory for importing, and press **ENTER** to read the files in the folder.

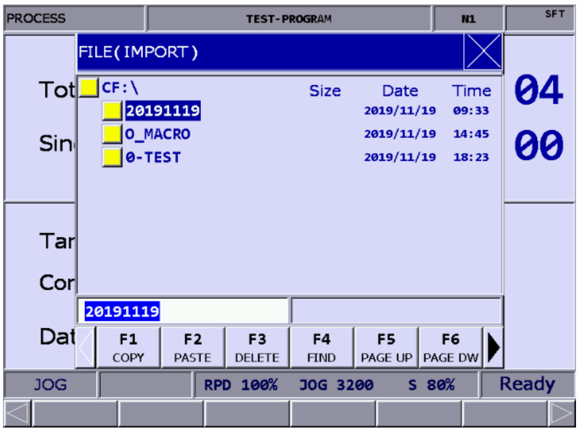


Figure 9.8.1

(4) After entering the parameter selection screen, press ⬆, ⬇, ⬅, and ➡ to move the cursor to the system parameter to be imported, and press **ENTER** to select or clear the check box.
(5) To select all the check boxes, press **SEL ALL**. To clear all the selected check boxes, press **CLR ALL**.

9

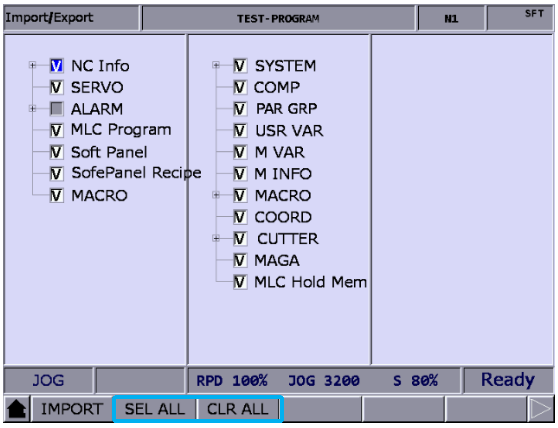


Figure 9.8.2

(6) Press **IMPORT** and a dialog box appears for confirmation. Enter “Y” and press **ENTER** to import the data of the files to the system. Then, the screen displays a progress bar showing the importing process. Cycle power to the system after completing importing the files.

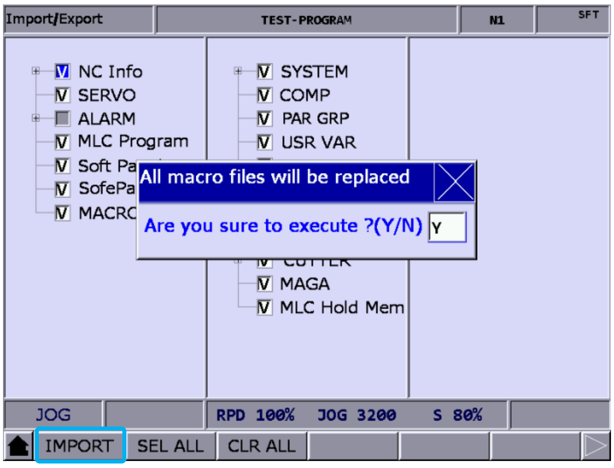


Figure 9.8.3

9.9 Export (EXPORT)

Use this function to back up the parameters of the system. The exported files are divided into three types: parameter files, MLC, and software panel. You need to have the proper authorization to use this function.

Type	Filename	Description
Parameter files	PAR.ncp	NC information, servo parameters, and alarms
MLC	MLC.gmc	MLC programs
MLC	MLC.lad	Image codes for MLC Ladder
MLC	MLC.lcm	Remarks for MLC Ladder
Software panel	HMI.cin	Screen information and element properties for software panel
Software panel	HMI.img	Graphic files for software panel
Software panel	HMI.sci	Project files for software panel

The operation steps for exporting parameters are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Press ▶ to display the function bar on the next page.

(3) Press **EXPORT** to enter the parameter selection screen.

(4) After entering the parameter selection screen, press ⬆, ⬇, ⬅, and ➡ to move the cursor to the system parameter to be exported, and press **ENTER** to select or clear the check box. To select all the check boxes, press **SEL ALL**. To clear all the selected check boxes, press **CLR ALL**.

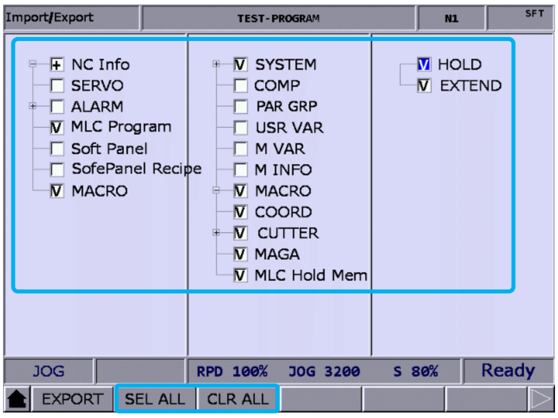


Figure 9.9.1

(5) Press **EXPORT** and the file manager (FILE) window appears as shown in Figure 9.9.2. Press ⬆ and ⬇ to select the directory for saving the exported files, or directly enter the folder name and press **ENTER** to save the exported files to the specified folder.

Figure 9.9.2

(6) Then, the screen displays a progress bar showing the exporting process until the exporting is finished.

(7) If you want to save the exported file to a new created folder, name the folder, then press **FOLDER** to save the exported file in the folder, as shown in Figure 9.9.2.

(8) If you save the exported file to a folder where an exported file already exists, a confirmation window appears as shown in Figure 9.9.3. Enter "Y" and press **ENTER** to replace the existing file with the newly exported file.

9

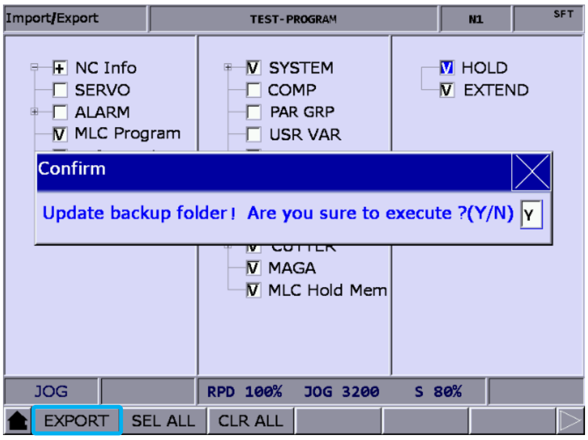


Figure 9.9.3

9.10 Multi-language (TEXT WR)

The default display languages for the group screens and corresponding function bars are Traditional Chinese, Simplified Chinese, and English. If there is a need for other languages, use this function to switch the interface to other languages. Contact the distributors or equipment suppliers for related information.

9.11 LOGO (LOGO WR)

Use this function to set the startup screen of the NC system with user-defined pictures for displaying the trademark or for other purposes. This function is available only when you have the proper authorization. The operation steps are as follows.

(1) Press **DGN** to enter the DGN screen.

(2) Repeatedly press ▶ to display the function bar on the last page.

(3) Insert the USB disk with the correct system startup screen file (SYSLOGO.bin) in it to the controller.

(4) Press **LOGO WR** and a dialog box appears for confirmation.

(5) Enter “Y”, press **OK**, and the system automatically reads and loads the startup screen file from the USB disk.

(6) After the file is updated, restart the system to display the updated startup screen.

Alarm (ALM) Group 10

The ALM group displays the alarm messages issued by the system in real time.

10

If an alarm occurs due to execution error or incorrect command format, the Alarm screen is automatically displayed. This group shows the alarm messages issued by the system in real time for you to troubleshoot the errors according to the displayed alarm information. In addition to displaying the current alarms, the ALM group also records the previous alarms.

Note: bold function names in a box (such as **POS**) mean the keys on machine operation panel A;
bold function names (such as **CLR ALL**) mean the function keys of F1 - F6.

10.1 Alarm (ALARM)

When an alarm occurs, troubleshoot the issue first, and then press **RESET** to clear the alarm and set the system to the initial status. The alarm display screen is as shown in the following figure and the sections with indicators show information about the alarms.

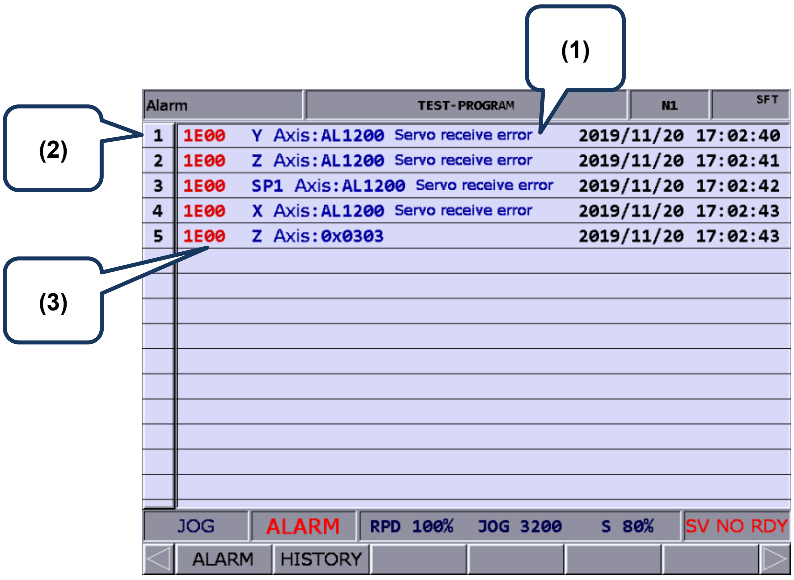


Figure 10.1.1

(1) Alarm message

(2) Sequence of alarm occurrence

(3) Alarm code

The operation steps for displaying and clearing the alarm messages are as follows.

(1) Press **ALM** to enter the ALM screen.

(2) Press **ALARM** to enter the alarm message screen.

(3) Press **RESET** to clear the alarm messages shown on the screen.

10.2 Alarm history (HISTORY)

This function records all the issued alarm messages and history information. In the History screen, you can access the error history during system execution as well as troubleshoot and analyze the errors according to the occurrence time and types of alarms. The alarm history records the occurrence time and names of the alarms. It can record up to 512 sets of data.

In addition to displaying the alarm information, you can also delete the alarm history with this function.

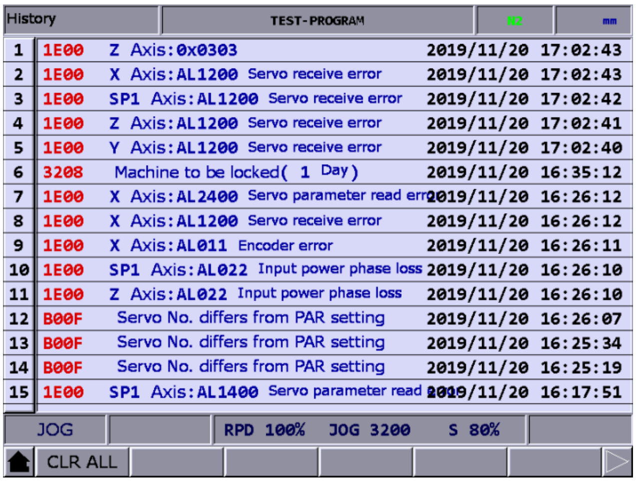


Figure 10.2.1

The operation steps for clearing all the alarm history are as follows.

(1) Press **ALM** to enter the ALM screen.

(2) Press **HISTORY** to enter the alarm history screen.

(3) Press **CLR ALL** and a confirmation window appears.

(4) Press **Y** and then **ENTER** to clear all the alarm history.

(This page is intentionally left blank.)

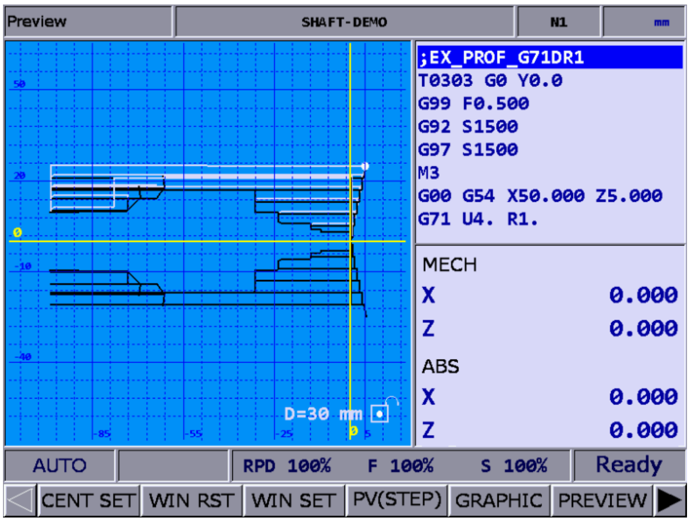
Graph (GRA) Group 11

The GRA group displays the real-time motion trajectory when the program is executing or checks the machining program when the program is not executing.

The GRA group provides two functions, trajectory display and program simulation.

Trajectory display: displays the real-time motion trajectory during machining.

Program simulation: checks the accuracy of the program format and machining path before machining.



Note: bold function names in a box (such as **POS**) mean the keys on machine operation panel A; bold function names (such as **CLR ALL**) mean the function keys of F1 - F6.

11.1 Trajectory display

When a machining program is executing, switch the system to GRA mode, and the system automatically draws the motion trajectory of the current program on the screen, with which you can check if the actual machining path is correct.

Function descriptions of the keys are as follows.

(1) **GRA**: enters the GRA screen.

(2) **CENT SET**: displays the current motion trajectory in the center of the display area.

(3) **WIN RST**: scales the graph to a moderate size and locates it in the middle of the screen.

(4) **WIN SET**: press **UP**, **DOWN**, **LEFT**, **RIGHT**, **ZM IN**, and **ZM OUT** in the WIN SET screen to adjust the display of the path graph.

(5) **CLEAR**: clears the content in the display area.

11.2 Program simulation (Preview)

This function draws the complete path of the G-code program for you to check if the program format is correct before the machining starts. This function can be further divided into GRAPHIC and PREVIEW.

Function descriptions of the keys are as follows.

(1) **GRA**: enters the GRA screen.

(2) **CENT SET**: displays the current motion trajectory in the center of the display area.

(3) **WIN RST**: scales the graph to a moderate size and locates it in the middle of the screen.

(4) **WIN SET**: press **UP**, **DOWN**, **LEFT**, **RIGHT**, **ZM IN**, and **ZM OUT** in the WIN SET screen to adjust the display of the path graph.

(5) **PV(STEP)**: draws the path of a single block for each press of the key.

(6) **GRAPHIC**: checks the program format without referring to the software limit and draws the complete path according to the program.

(7) **PREVIEW**: checks the program format by referring to the software limit. The settings of the coordinate system and the tool compensation should match the actual application so the complete path can be correctly drawn.

(8) **CLEAR**: clears the content in the display area.

Pay attention to the following when using the functions of GRA:

- When the Preview function is enabled, machining execution is prohibited.
- When the Preview function is enabled, switching the system mode will force close this function.
- If you cancel the preview during previewing, you need to start from the initial block if desiring to preview again.
- Graphs drawn by the function of GRAPHIC or PREVIEW may exceed the display area because of the setting values of the workpiece coordinate. If so, press **GRAPHIC** or **PREVIEW** again, and the system automatically displays a moderate preview graph in the center of the display area.
- The Preview function in the lathe system only displays graphs from the angle of view of the X-Z plane.
- Use the graphic parameter Pr14003 to set the default display of the graph.

Parameter (PAR) Group 12

This chapter introduces functions and settings for all of the parameters.

Parameters in the PAR group can be divided into operation parameters, tool magazine parameters, machining parameters, spindle parameters, machine parameters, homing parameters, compensation parameters, and system parameters according to their functions. The timing for parameters to take effect is divided into three types according to their properties: after cycling power to the servo drive (S), after cycling power to the system (P), or after pressing **RESET** (R).

Note:

1. Bold function names in a box (such as **POS**) mean the keys on machine operation panel A; bold function names (such as **CLR ALL**) mean the function keys of F1 - F6.
2. In the PAR screen, enter “S + parameter number” and you are directed to the specified parameter immediately.

12.1 Machining parameter (PROCESS)

Machining parameters such as maximum cutting speed, and cutting and smoothing acceleration and deceleration time can have prominent effects on the machining results. Properly set the values according to the actual requirements to achieve the best performance. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press **PROCESS** to enter the machining parameter setting screen.

(3) Press ↑ and ↓ to move the cursor to the specified field, and enter a value within the range specified in the lower right corner of the screen as shown in Figure 12.1.1.

(4) Press **ENTER** to complete the setting.

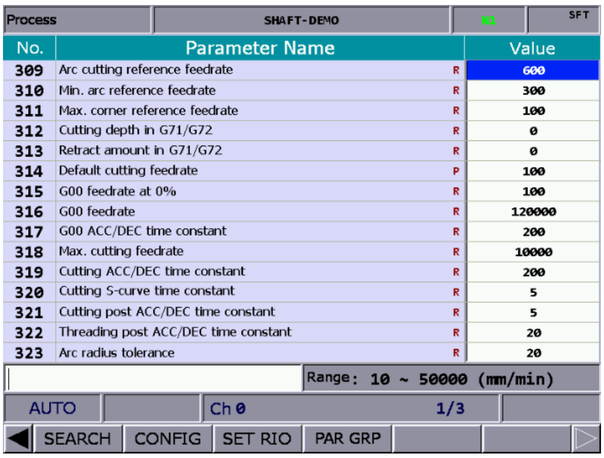
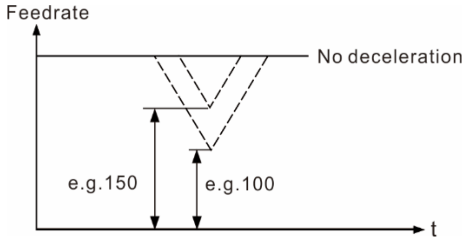
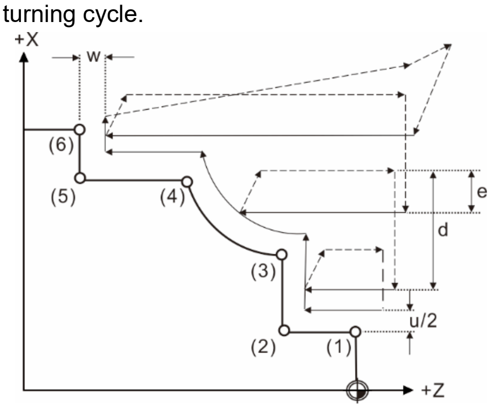
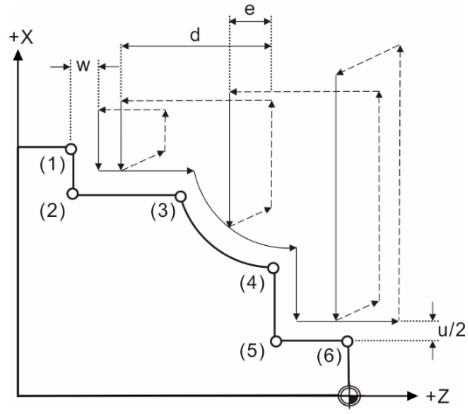


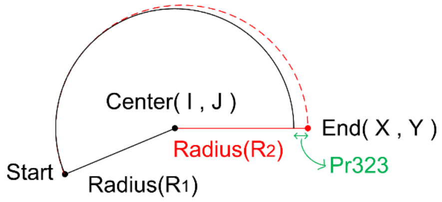
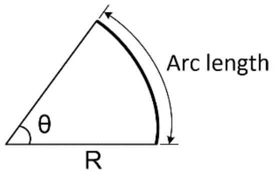
Figure 12.1.1

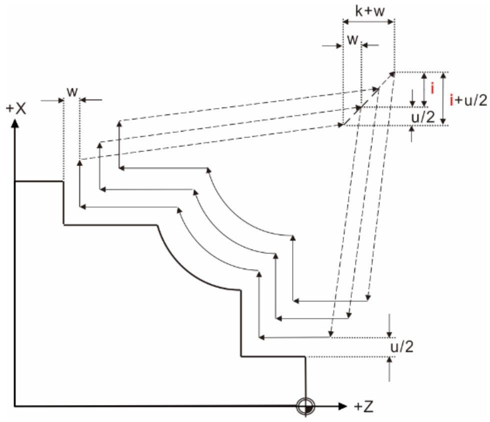
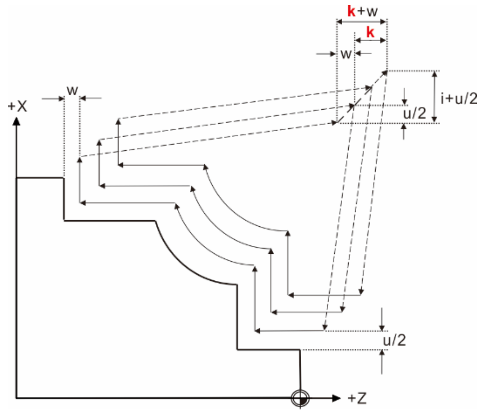
12.1.1 Machining parameter descriptions

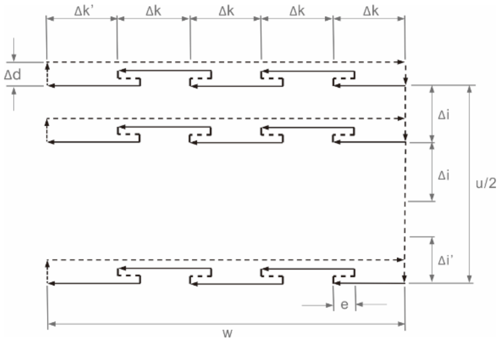
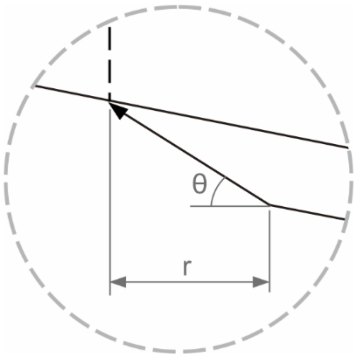
Para-meter No.	Item	Description	Default value	Setting range	Note
309	Arc cutting reference feedrate	During circular interpolation, the arc path shifts inward because of the delayed following of the servo. With the shift amount fixed, you can set the maximum feedrate of the arc radius with this parameter. The higher the parameter value, the greater the shift amount and the lower the machining precision, and vice versa. Unit: mm/min	1000	10 - 50000	R
310	Min. arc reference feedrate	Sets the minimum feedrate for executing circular interpolation. The higher the feedrate, the less precise the machining and the more the contouring error, while it shortens the machining time. Unit: mm/min	500	10 - 50000	R
311	Max. corner reference feedrate	Sets the speed limit at corners. The higher the value, the easier to keep the high speed at corners for restoring to the feedrate before deceleration. Setting the value too high or improper setting may cause vibration of the machine at the corners. Unit: mm/min 	100	0 - 50000	R
312	Cutting depth in G71/G72	The default cutting depth for executing G71/G72 turning cycle. Refer to the cutting depth d in the following path diagram of G71 turning cycle. 	1000	0 - 50000	R

Para-meter No.	Item	Description	Default value	Setting range	Note
313	Retract amount in G71/G72	The default retract amount for executing G71/G72 turning cycle. Refer to the retract amount e in the following path diagram of G71 turning cycle.  	1000	0 - 50000	R
314	Default cutting feedrate	Sets the default cutting speed. When you set this parameter without specifying the F value in the cutting command, the cutting speed is the value set by this parameter. Unit: mm/min, inch/min	0	0 - 20000	P
315	G00 feedrate at 0%	Sets the feedrate for rapid traverse when the rapid traverse override is set to 0%. Unit: mm/min, inch/min	100	10 - 10000	R
316	G00 feedrate	Sets the maximum speed for RAPID mode and G00. Unit: mm/min, inch/min	5000	1 - 60000	R
317	G00 ACC/DEC time constant	Sets the acceleration and deceleration time for rapid traverse. Unit: msec	200	1 - 2000	R
318	Max. cutting feedrate	Sets the maximum cutting feedrate. Unit: mm/min, inch/min	5000	1 - 60000	R
319	Cutting ACC/DEC time constant	Sets the acceleration and deceleration time for cutting speed. Unit: msec (acceleration and deceleration time before interpolation)	200	1 - 2000	R
320	Cutting S-curve time constant	Sets the S-curve time for cutting speed. Unit: msec (acceleration and deceleration time before interpolation)	20	1 - 2000	R
321	Cutting post ACC/DEC time constant	Sets the post acceleration and deceleration time for cutting speed. The higher the value, the more the contouring error. Unit: msec (acceleration and deceleration time after interpolation)	50	1 - 500	R
322	Threading post ACC/DEC time constant	The time required for accelerating or decelerating to the target speed when threading. Target speed = number of revolution / min x thread pitch The smaller the parameter value, the shorter the ineffective threads, but the vibration is more violent. The greater the parameter value, the longer the ineffective threads, but the vibration is less violent.	10	1 - 100	R

12

Para-meter No.	Item	Description	Default value	Setting range	Note
323	Arc radius tolerance	 When the circular path is specified in center format (I, J, K), the controller calculates the radius R_1 (the distance from the start point to the circle center) and the radius R_2 (the distance from the end point to the circle center). The absolute difference between R_1 and R_2 should be smaller than Pr323 ($\|R_1-R_2\| <$ Pr323), or alarm B00D Radius mismatch occurs. Unit: μm	1	1 - 60000	R
329	Max. block length of path smoothing	When the system performs curve fitting, if the block length of G01 exceeds the setting in Pr329, the curve fitting for the block is automatically canceled and the linear interpolation remains effective. Unit: 0.1 mm	20	0 - 10000	R
330	Min. corner angle of path smoothing	When the angle specified by a single block exceeds the angle of curve fitting, the curve fitting for the corner is automatically canceled and the sharpness is kept. Unit: degree	15	0 - 90	R
333	Contour tolerance of path smoothing	Cosine error for curve fitting. The greater the value, the smoother the curve. But if the tolerance is set too great, the more the contouring error. The smaller the value, the more likely the machining contour is close to the programmed path for linear interpolation, but the curve is less smooth. It is suggested that the parameter value and the error value set in the CAM software should be consistent so as to make the path smoother without affecting the precision. Unit: 0.0001 mm	100	0 - 50000	R
344	Radius of rotation axis	Sets the radius for the rotation axis. The greater the value, the slower the rotation speed and vice versa. When you set Pr344 to 0, this function is disabled and the rotation axis feedrate is determined by the F value (deg/min). When you set Pr344 to a non-zero value, the system defines this value as the radius of the rotation axis to calculate the arc length, and performs interpolation based on the arc length. Meanwhile, the feedrate for the rotation axis is determined by the F value (mm/min).  $Arc\ \ length\ \ (mm) = \frac{\theta}{180}\ \ R\pi$	0	0 - 65535	R

Para-meter No.	Item	Description	Default value	Setting range	Note
344	Radius of rotation axis	The default unit of feedrate for linear axes is mm/min and that for the rotation axis is deg/min. If the rotation axis used in the application requires the feedrate to be consistent with the tangential velocity (mm/min), you can set Pr344 with the setting value as close as the distance between the cutting point and the rotation center (rotation radius). The smaller the value of Pr344, the faster the cutting speed; the greater the value of Pr344, the slower the cutting speed. Unit: 0.1 mm	0	0 - 65535	R
345	X axis cutting depth in G73	Default cutting amount in X-axis direction for executing G73 turning cycle. Refer to the following G73 turning cycle path for the cutting amount i in X-axis direction. 	1000	0 - 50000	R
346	Z axis cutting depth in G73	Default cutting amount in Z-axis direction for executing G73 turning cycle. Refer to the following G73 turning path for the cutting amount k in Z-axis direction. 	1000	0 - 50000	R
347	Number of cutting times in G73	Default cutting times for executing G73 turning cycle.	3	1 - 99	R

Para-meter No.	Item	Description	Default value	Setting range	Note
348	Retract amount in G74/G75	The default retract amount e for executing G74/G75 peck turning cycle. 	1000	0 - 50000	R
349	Chamfering angle in G76/G92 threading	Chamfering angle θ for executing G76/G92 thread turning cycle. 	45	1 - 89	R
380	Chamfering length in G76/G92 threading	Chamfering length r for executing G76/G92 thread turning cycle. Chamfering length = parameter value x (0.1 x thread pitch). Assume the pitch is L, then the chamfering length for thread turning can be 0L - 12.7L. (refer to the figure in Pr349 column for the length of r)	3	0 - 127	R
381	Finishing count in G76 threading	Finishing count for executing G76 thread turning cycle.	1	1 - 99	R
382	Tool nose angle in G76 threading	Tool nose angle for executing G76 thread turning cycle.	60	0 - 80	R
383	Min. cutting depth in G76 threading	Minimum cutting depth for executing G76 thread turning cycle.	1000	0 - 50000	R
510	Block No. to check in tool comp. interference	Sets the number of blocks to check in tool compensation interference: Pr510 = 0: number of blocks = 3 Pr510 < 3: number of blocks = Pr510	0	0 - 3	P
511	G00 S-curve time constant	Sets the acceleration and deceleration S-curve for the G00 movement. Unit: msec	1	0 - 2000	R
515	Finishing allowance in G76 threading	Finishing allowance for executing G76 thread turning cycle.	200	0 - 50000	R

12.2 Operating parameter (OPERATE)

You can use macro programs to perform mathematical operations, logical statement, and repeated program call to increase flexibility in program editing for G-code execution. You can set whether to execute specified macro programs in the Operation screen. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press **OPERATE** to enter the operating parameter setting screen.

(3) Press ↑, ↓, ←, and → to move the cursor to the specified field and enter a valid value within the range displayed in the lower right corner of the screen as shown in Figure 12.2.1.

(4) Press **ENTER** to complete the setting.

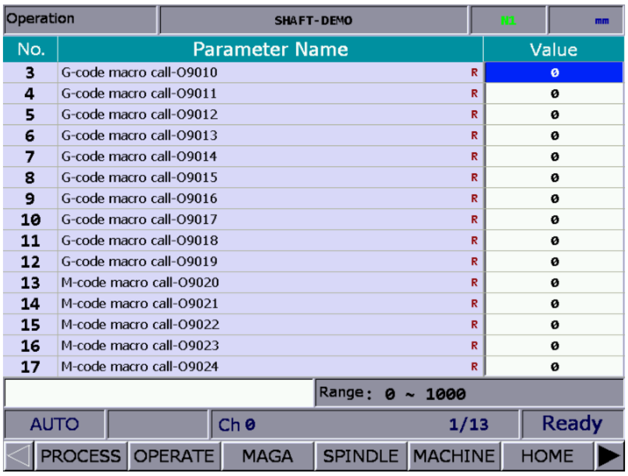


Figure 12.2.1

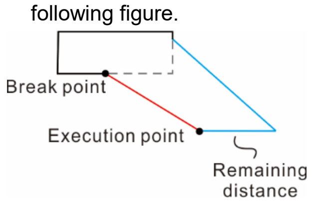
12.2.1 Operating parameter descriptions

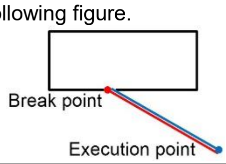
<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td rowspan="10">3 - 12</td><td rowspan="10">G-code macro call - O9010 to O9019</td><td>G-code macro call-O9010
Set these parameters to 0 to disable macro call.</td><td rowspan="10">0</td><td rowspan="10">0 - 1000</td><td rowspan="10">R</td></tr>
<tr><td>G-code macro call-O9011</td></tr>
<tr><td>G-code macro call-O9012</td></tr>
<tr><td>G-code macro call-O9013</td></tr>
<tr><td>G-code macro call-O9014</td></tr>
<tr><td>G-code macro call-O9015</td></tr>
<tr><td>G-code macro call-O9016</td></tr>
<tr><td>G-code macro call-O9017</td></tr>
<tr><td>G-code macro call-O9018</td></tr>
<tr><td>G-code macro call-O9019</td></tr>
<tr><td rowspan="10">13 - 22</td><td rowspan="10">M-code macro call - O9020 to O9029</td><td>M-code macro call-O9020
Set these parameters to 0 to disable macro call.</td><td rowspan="10">0</td><td rowspan="10">0 - 1000</td><td rowspan="10">R</td></tr>
<tr><td>M-code macro call-O9021</td></tr>
<tr><td>M-code macro call-O9022</td></tr>
<tr><td>M-code macro call-O9023</td></tr>
<tr><td>M-code macro call-O9024</td></tr>
<tr><td>M-code macro call-O9025</td></tr>
<tr><td>M-code macro call-O9026</td></tr>
<tr><td>M-code macro call-O9027</td></tr>
<tr><td>M-code macro call-O9028</td></tr>
<tr><td>M-code macro call-O9029</td></tr>
<tr><td>23</td><td>T-code macro call - O9000</td><td>T-code macro call-O9000
0: off; 1: on</td><td>0</td><td>0 - 1</td><td>R</td></tr>
<tr><td>24</td><td>Call O9030 after break line search</td><td>Call O9030 after break line search.
0: function disabled. After finding the break line, the system continues executing the program without calling a macro.
1: function enabled. After finding the break line, the system first calls and executes O9030 once CYCLE START is pressed, and then it returns to the main program and carries on the machining program followed by the breakpoint.</td><td>0</td><td>0 - 1</td><td>R</td></tr>
<tr><td>25</td><td>System DIO setting</td><td>Sets G31 input polarity.
0: NC; 1: NO<table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>G31 high speed input point 1</td><td>0 - 1</td></tr><tr><td>1</td><td>G31 high speed input point 2</td><td>0 - 1</td></tr></table></td><td>0</td><td>0 - 1</td><td>P</td></tr>
<tr><td>46</td><td>System application setting</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0 - 1</td><td>Output control mode
0: DMCNET</td><td>0</td></tr><tr><td>2</td><td>Homing before machining
0: Y; 1: N</td><td>0 - 1</td></tr><tr><td>4</td><td>Screw unit
0: metric; 1: imperial</td><td>0 - 1</td></tr><tr><td>5</td><td>G31 high speed input 1 switch
0: off; 1: on</td><td>0 - 1</td></tr></table></td><td>96</td><td>0 - 0xFFFF</td><td>P</td></tr>
</table>

<table>
<tr><th>Para-meter No.</th><th>Item</th><th colspan="3">Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td rowspan="10">46</td><td rowspan="10">System application setting</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="10">96</td><td rowspan="10">0 - 0xFFFF</td><td rowspan="10">P</td></tr>
<tr><td>6</td><td>G31 high speed input 2 switch
0: off; 1: on</td><td>0 - 1</td></tr>
<tr><td>7</td><td>Hardware limit check
0: on; 1: off</td><td>0 - 1</td></tr>
<tr><td>8</td><td>Software limit check
0: on; 1: off</td><td>0 - 1</td></tr>
<tr><td>10</td><td>Omit decimal places of the floating point number in the motion command
0: do not omit
(example: input 1 to indicate 1 μm)
1: omit
(example: input 1 to indicate 1 mm)</td><td>0 - 1</td></tr>
<tr><td>11</td><td>G00 operation mode
0: multiple axes perform synchronous interpolation and reach the positioning point at the same time
1: each axis performs individual interpolation and reaches the positioning point separately at different speeds</td><td>0 - 1</td></tr>
<tr><td>12</td><td>Macro look-ahead
0: off; 1: on</td><td>0 - 1</td></tr>
<tr><td>13</td><td>G00 path blending mode
0: same axis
1: different axes
The greater the setting value of Pr334, the larger the chamfer angle of the path.</td><td>0 - 1</td></tr>
<tr><td>14</td><td>Return mode of one-key macro call
0: go to the next line
1: return to the interrupted line</td><td>0 - 1</td></tr>
<tr><td>15</td><td>Pre-warning for software limit
0: off; 1: on</td><td>0 - 1</td></tr>
<tr><td>47</td><td>MGP gain</td><td colspan="3">MPG filter gain. For setting the MPG response. The greater the value, the faster the response, but the machine is subject to vibration.
Unit: 0.0001</td><td>100</td><td>1 - 60000</td><td>R</td></tr>
<tr><td>48</td><td>MPG filter</td><td colspan="3">Sets the MPG filter level.
0: disabled<table><tr><td>Level</td><td>1</td><td>2</td><td>3</td><td>4</td><td>5</td><td>6</td></tr><tr><td>kHz</td><td>31</td><td>10</td><td>5</td><td>2.5</td><td>1.6</td><td>1.2</td></tr></table></td><td>0</td><td>0 - 6</td><td>R</td></tr>
<tr><td>49</td><td>Axis port input setting</td><td colspan="3">0: the 4th axis include both limit and origin signals
1: changes the positive limit signal, negative limit signal, and origin signal of the 4th axis to the origin signals of the 4th, 5th, and 6th axes.</td><td>0</td><td>0 - 1</td><td>R</td></tr>
<tr><td rowspan="3">50</td><td rowspan="3">Macro file display</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="3">0</td><td rowspan="3">0 - 3</td><td rowspan="3">-</td></tr>
<tr><td>0</td><td>Display O macro file</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Display G/M macro file</td><td>0 - 1</td></tr>
</table>

12

<table>
<tr><th>Para-meter No.</th><th>Item</th><th colspan="3">Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td rowspan="15">51</td><td rowspan="15">System auxiliary tool</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="15">0</td><td rowspan="15">0 - 1</td><td rowspan="15">P</td></tr>
<tr><td>0</td><td>Spindle check before cutting: when this function is enabled, if a cutting command is executed but the spindle is not in operation, the system displays an alarm.
0: off; 1: on</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Non-volatile setting for #10450 - #10500
0: on; 1: off</td><td>0 - 1</td></tr>
<tr><td>4</td><td>G54 - G59 offset coordinate switch
0: off; 1: on</td><td>0 - 1</td></tr>
<tr><td>5</td><td>Stroke protection
0: off; 1: on</td><td>0 - 1</td></tr>
<tr><td>6</td><td>1st spindle OA/OB signal sequence
0: AB; 1: BA</td><td>0 - 1</td></tr>
<tr><td>8</td><td>Spindle speed percentage reference during threading
0: set percentage; 1: 100%</td><td>0 - 1</td></tr>
<tr><td>9</td><td>Spindle target speed check during cutting
0: off; 1: on</td><td>0 - 1</td></tr>
<tr><td>10</td><td>Return mode after triggering M96
0: go to the next line;
1: return to the interrupted line</td><td>0 - 1</td></tr>
<tr><td>11</td><td>Tapping mode
0: open-loop; 1: following</td><td>0 - 1</td></tr>
<tr><td>12</td><td>G98 / G99 default setting
0: program; 1: Pr306</td><td>0 - 1</td></tr>
<tr><td>14</td><td>MPG reverse function
0: off; 1: on</td><td>0 - 1</td></tr>
<tr><td>15</td><td>Feedback check
0: on; 1: off</td><td>0 - 1</td></tr>
<tr></tr><tr></tr>
<tr><td>301</td><td>Unit decimal places</td><td colspan="3">Unit setting for coordinate display. For example, when you set this parameter to 3, the coordinates are displayed with three decimal places, such as -99999.999 to 99999.999.</td><td>3</td><td>0 - 4</td><td>P</td></tr>
<tr><td rowspan="4">305</td><td rowspan="4">Channel auxiliary setting</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="4">0</td><td rowspan="4">362</td><td rowspan="4">P</td></tr>
<tr><td>5</td><td>EMG stop source
0: system; 1: M1079</td><td>0 - 1</td></tr>
<tr><td>6</td><td>G00 / G01 transition speed
0: to zero speed
1: no deceleration (refers to Pr334)</td><td>0 - 1</td></tr>
<tr><td>8</td><td>Record machine coordinates after triggering HSI
0: command; 1: feedback</td><td>0 - 1</td></tr>
<tr><td rowspan="4">306</td><td rowspan="4">G-code programming parameter</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="4">532</td><td rowspan="4">0 - 0xFFFF</td><td rowspan="4">P</td></tr>
<tr><td>0</td><td>Default unit
0: metric; 1: imperial</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Default programming mode
0: absolute; 1: incremental</td><td>0 - 1</td></tr>
<tr><td>2</td><td>Default feed mode
0: feed/min; 1: feed/rev</td><td>0 - 1</td></tr>
</table>

<table>
<tr><th>Para-meter No.</th><th>Item</th><th colspan="3">Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td rowspan="5">306</td><td rowspan="5">G-code programming parameter</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="5">532</td><td rowspan="5">0 - 0xFFFF</td><td rowspan="5">P</td></tr>
<tr><td>4 - 5</td><td>Default plane
0: G17; 1: G18; 2: G19</td><td>0 - 2</td></tr>
<tr><td>9 - 10</td><td>Lathe G-code type
0: A; 1: B; 2: C</td><td>0 - 3</td></tr>
<tr><td>11</td><td>Set the X axis to diameter / raidus mode
0: diameter; 1: radius</td><td>0 - 1</td></tr>
<tr><td>13 - 15</td><td>Default workpiece coordinates
0 - 5: G54 - G59</td><td>0 - 5</td></tr>
<tr><td rowspan="7">307</td><td rowspan="7">Channel application setting</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="7">0xD4</td><td rowspan="7">0 - 0xFFFF</td><td rowspan="7">P</td></tr>
<tr><td>0</td><td>Tool length compensation mode
0: when the block has a tool length compensation execution or cancellation command without a Z-axis command, Z axis moves.
1: when the block has a tool length compensation execution or cancellation command without a Z-axis command, Z axis does not move.</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Spindle speed after reset
0: command speed remains after reset
1: command speed becomes 0 after reset</td><td>0 - 1</td></tr>
<tr><td>2</td><td>Software limit check mode
0: program path; 1: tool path (compensated)</td><td>0 - 1</td></tr>
<tr><td>3</td><td>Tool length input mode
0: absolute; 1: incremental</td><td>0 - 1</td></tr>
<tr><td>4 - 5</td><td>G31 input source
0; off; 1: HSI 1 (latch input 1);
2: HSI 2 (latch input 2);
3: HSI 1 & 2</td><td>0 - 1</td></tr>
<tr><td>6</td><td>Running mode after interruption
0: during the execution of a block, if you manually move the tool from its original position and then resumes the program execution, the tool moves the remaining distance first and then returns to the original path in the next motion block, as shown in the following figure.


1: during the execution of a block, if you manually move the tool from its original position and then resumes the program execution, the tool returns to the original path first and then moves the remaining distance, as shown in the following figure.

</td><td>0 - 1</td></tr>
</table>

Parameter No.	Item	Bit	Function	Range	Default value	Setting range	Note
307	Channel application setting	8 - 9	EMG stop mode 0: after the controller stops, the servo switches to Servo Off 1: after the controller stops, the servo decelerates to a stop and switches to Servo Off 2: after the controller stops, the servo decelerates to a stop and remains Servo On	0 - 2	0xD4	0 - 0xFFFF	P
		10	MLC variable type 0: word type 1: double word type Set this bit to 1 to change the interface input / output registers to 8 sets (32-bit) from 16 sets (16-bit).	0 - 1			
		11	G00/G01 max. feedrate reference 0: resultant velocity G00: Pr316 & Pr621 G01: Pr318 & Pr625 1: individual velocity G00: Pr621 G01: Pr625	0 - 1			
		12	Return mode after interruption 0: G00; 1: G01 You can use this parameter to restore the motion mode when the machining was interrupted in the following condition: you press **FEED HOLD** during machining, set the system to MDI mode, move the machine, and then set the system back to AUTO mode and resume the machining without resetting.	0 - 1			
		14	Synchronous execution for M / S / T-code and G00 0: off; 1: on	0 - 1			
		15	Cancel tool radius compensation for G00 block 0: off; 1: on	0 - 1			
308	Channel auxiliary setting	0	Feedrate reference 0: program; 1: Pr314	0 - 1	0	0 - 0xFFFF	P
		1	Local variable NULL function 0: off, #1 - #50 are 0 1: on, #1 - #50 are null	0 - 1			
		2	Current monitoring function 0: off; 1: on	0 - 1			
		4	Tool compensation interference check 0: on; 1: off	0 - 1			
		5	Motion speed remains when coordinates switch 0: off; 1: on	0 - 1			
		6	Motion speed remains when macro call 0: off; 1: on	0 - 1			
		9	Pause before running M99 0: off; 1: on	0 - 1			
		11	ABS coordinate changes after tool length / wear modification 0: remain; 1: change	0 - 1			
		12	Tool length compensation after M30 / M02 / RST / Bootup 0: cancel; 1: maintain	0 - 1			
		13	Tool number display mode 0: G-code; 1: D1115	0 - 1			

<table>
<tr><th>Para-
meter
No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>308</td><td>Channel auxiliary setting</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>14</td><td>Feed/rev mode reference
0: command; 1: feedback</td><td>0 - 1</td></tr><tr><td>15</td><td>C axis mode
0: turning; 1: lathe</td><td>0 - 1</td></tr></table></td><td>0</td><td>0 - 0xFFFF</td><td>P</td></tr>
<tr><td>324</td><td>Peck-drilling escape amount</td><td>Sets the retract amount for performing peck drilling. Unit: μm</td><td>100</td><td>1 - 50000</td><td>R</td></tr>
<tr><td>326</td><td>Cycle parameter</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0 - 1</td><td>Tool withdraw direction (for G76 / G87 cycle command)
0: +X; 1: -X; 2: +Y; 3: -Y</td><td>0 - 3</td></tr><tr><td>2 - 3</td><td>Drilling / tapping mode
0: general
1: deep-pecking (feed amount = Q, retract amount = R)
2: pecking (feed amount = Q, retract amount = D)
3: deep-pecking (feed amount = Q, retract amount = R, dwell time = Pr513)</td><td>0 - 3</td></tr></table></td><td>0</td><td>0 - 0xFFFF</td><td>R</td></tr>
<tr><td>327</td><td>EMG stop time constant</td><td>Sets the time for the servo motor to decelerate to zero speed when the emergency stop is pressed in AUTO mode. Unit: msec</td><td>50</td><td>5 - 500</td><td>R</td></tr>
<tr><td>328</td><td>EMG stop delay time</td><td>Sets the delay time for the special M relay M2114 System emergency stop when the system is stopped and in Servo Off status. Unit: msec</td><td>35</td><td>0 - 2000</td><td>R</td></tr>
<tr><td>334</td><td>G00 blending ratio</td><td>The speed does not decelerate to zero when G00 is executed between blocks. Use this parameter to set the blending ratio. The greater the value, the less the deceleration at block transitions and the faster the speed; the smaller the value, the more the deceleration at block transitions and the slower the speed. Unit: %</td><td>0</td><td>0 - 100</td><td>R</td></tr>
<tr><td rowspan="8">350 - 357</td><td rowspan="8">Halt M-code 1 - 8</td><td>Halt M-code 1
0: disabled</td><td rowspan="8">0</td><td rowspan="8">0 - 1000</td><td rowspan="8">P</td></tr>
<tr><td>Halt M-code 2</td></tr>
<tr><td>Halt M-code 3</td></tr>
<tr><td>Halt M-code 4</td></tr>
<tr><td>Halt M-code 5</td></tr>
<tr><td>Halt M-code 6</td></tr>
<tr><td>Halt M-code 7</td></tr>
<tr><td>Halt M-code 8</td></tr>
<tr><td>358</td><td>Spindle > C axis mode switch M-code (lathe)</td><td>M-code for switching the spindle to C axis for the lathe system. This M-code can be used as a halt M-code.</td><td>0</td><td>0 - 1000</td><td>P</td></tr>
<tr><td>359</td><td>C axis > Spindle mode switch M-code (lathe)</td><td>M-code for switching the C axis to spindle for the lathe system. This M-code can be used as a halt M-code.</td><td>0</td><td>0 - 1000</td><td>P</td></tr>
</table>

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>360</td><td>Synchronization direction control</td><td>Sets the synchronization direction.
Bit 0 - 5: synchronous control of X - C axes
0: same direction
1: different directions
<table>
<tr><th>Bit</th><th>Function</th><th>Range</th></tr>
<tr><td>0</td><td>Synchronous direction for X axis</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Synchronous direction for Y axis</td><td>0 - 1</td></tr>
<tr><td>2</td><td>Synchronous direction for Z axis</td><td>0 - 1</td></tr>
<tr><td>3</td><td>Synchronous direction for A axis</td><td>0 - 1</td></tr>
<tr><td>4</td><td>Synchronous direction for B axis</td><td>0 - 1</td></tr>
<tr><td>5</td><td>Synchronous direction for C axis</td><td>0 - 1</td></tr>
</table></td><td>0</td><td>0 - 0x3F</td><td>P</td></tr>
<tr><td>361</td><td>Synchronous control X</td><td>Specifies the master axis when the X axis is the slave axis. For example, set this parameter to 2 if desiring to set the Y axis as the master axis for synchronous control.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>362</td><td>Synchronous control Y</td><td>Specifies the master axis when the Y axis is the slave axis.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>363</td><td>Synchronous control Z</td><td>Specifies the master axis when the Z axis is the slave axis.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>364</td><td>Synchronous control A</td><td>Specifies the master axis when the A axis is the slave axis.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>365</td><td>Synchronous control B</td><td>Specifies the master axis when the B axis is the slave axis.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>366</td><td>Synchronous control C</td><td>Specifies the master axis when the C axis is the slave axis.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
</table>

<table>
<tr><th>Para-
meter
No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>370</td><td>Transfer control direction</td><td>Sets the transfer direction.
Bit 0 - 5: transfer direction of X - C axes
0: same direction
1: different directions
<table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>Transfer direction X</td><td>0 - 1</td></tr><tr><td>1</td><td>Transfer direction Y</td><td>0 - 1</td></tr><tr><td>2</td><td>Transfer direction Z</td><td>0 - 1</td></tr><tr><td>3</td><td>Transfer direction A</td><td>0 - 1</td></tr><tr><td>4</td><td>Transfer direction B</td><td>0 - 1</td></tr><tr><td>5</td><td>Transfer direction C</td><td>0 - 1</td></tr></table></td><td>0</td><td>0 - 0x3F</td><td>P</td></tr>
<tr><td>371</td><td>Transfer control X</td><td>Specifies the X axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the X axis move while the originally commanded axis does not move. For example, set this parameter to 2 if desiring to transfer the control command from the Y axis.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>372</td><td>Transfer control Y</td><td>Specifies the Y axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the Y axis move while the originally commanded axis does not move.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>373</td><td>Transfer control Z</td><td>Specifies the Z axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the Z axis move while the originally commanded axis does not move.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>374</td><td>Transfer control A</td><td>Specifies the A axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the A axis move while the originally commanded axis does not move.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>375</td><td>Transfer control B</td><td>Specifies the B axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the B axis move while the originally commanded axis does not move.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>376</td><td>Transfer control C</td><td>Specifies the C axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the C axis move while the originally commanded axis does not move.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
</table>

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>508</td><td>Polygon cutting axis setting</td><td>The polygon cutting operation requires one spindle and one rotation axis. Use this parameter to set the rotation axis (tool axis) to use.
0: off; 4 - 6: A - C</td><td>0</td><td>0, 4 - 6</td><td>P</td></tr>
<tr><td>509</td><td>Torque limit for each axis</td><td>Torque limit for each servo axis; Bit 0 indicates X axis, Bit 1 indicates Y axis, Bit 2 indicates Z axis, and so on. For example, to enable the torque limit of X and Z axes, set Pr509 to 5.</td><td>0</td><td>0 - 65535</td><td>P</td></tr>
<tr><td>513</td><td>Drilling/tapping cycle dwell time above hole bottom</td><td>In drilling / tapping mode, set Pr326 to 3 and refer to the setting of this parameter as the dwell time at the top of the hole.</td><td>0</td><td>0 - 50000</td><td>R</td></tr>
<tr><td>514</td><td>Feedback check time</td><td>After the system issues a command, when the time set by Pr514 passes while the feedback value remains the same, alarm 4FFD occurs.
Unit: ms</td><td>0</td><td>0 - 65535</td><td>R</td></tr>
<tr><td>2006</td><td>Input source setup</td><td><table>
<tr><th>Bit</th><th>Function</th><th>Range</th></tr>
<tr><td>14</td><td>Origin / limit planning
0: off; 1: on</td><td>0 - 1</td></tr>
<tr><td>15</td><td>EMG stop source
0: IES
1: HSI on OPENCNC</td><td>0 - 1</td></tr>
</table></td><td>0</td><td>0 - 49152</td><td>P</td></tr>
<tr><td>2010</td><td>High speed input trigger setting</td><td><table>
<tr><th>Bit</th><th>Function</th><th>Range</th></tr>
<tr><td>0</td><td>HSI 1 trigger setting</td><td>0 - 1</td></tr>
<tr><td>1</td><td>HSI 2 trigger setting</td><td>0 - 1</td></tr>
</table>Sets to 0 means rising-edge triggering; sets to 1 means falling-edge triggering.</td><td>0</td><td>0 - 65535</td><td>P</td></tr>
<tr><td>2049</td><td>PMC module output setting</td><td>PMC module output setting
0: general (servo)
1: bridge (command = feedback)<table>
<tr><th>Bit</th><th>Function</th><th>Range</th></tr>
<tr><td>0</td><td>1st axis feedback source</td><td>0 - 1</td></tr>
<tr><td>1</td><td>2nd axis feedback source</td><td>0 - 1</td></tr>
<tr><td>2</td><td>3rd axis feedback source</td><td>0 - 1</td></tr>
<tr><td>3</td><td>4th axis feedback source</td><td>0 - 1</td></tr>
<tr><td>4</td><td>5th axis feedback source</td><td>0 - 1</td></tr>
<tr><td>5</td><td>6th axis feedback source</td><td>0 - 1</td></tr>
<tr><td>6</td><td>7th axis feedback source</td><td>0 - 1</td></tr>
<tr><td>7</td><td>8th axis feedback source</td><td>0 - 1</td></tr>
<tr><td>8</td><td>9th axis feedback source</td><td>0 - 1</td></tr>
<tr><td>9</td><td>10th axis feedback source</td><td>0 - 1</td></tr>
<tr><td>10</td><td>11th axis feedback source</td><td>0 - 1</td></tr>
<tr><td>11</td><td>12th axis feedback source</td><td>11</td></tr>
<tr><td>12 - 13</td><td>Pulse command type
0: A/B phase
1: CW/CCW
2: Pulse/direction</td><td>0 - 2</td></tr>
<tr><td>14</td><td>Pulse logic
0: positive; 1: negative</td><td>0 - 1</td></tr>
</table></td><td>0</td><td>0 - 65535</td><td>P</td></tr>
</table>

Para-meter No.	Item	Description	Default value	Setting range	Note
2050	1st PMC encoder resolution (single-phase)	The motor single-frequency pulse number.	0	0 - 65535	P
2051	2nd PMC encoder resolution (single-phase)	The motor single-frequency pulse number.	0	0 - 65535	P
2052	3rd PMC encoder resolution (single-phase)	The motor single-frequency pulse number.	0	0 - 65535	P
2053	4th PMC encoder resolution (single-phase)	The motor single-frequency pulse number.	0	0 - 65535	P
2054	5th PMC encoder resolution (single-phase)	The motor single-frequency pulse number.	0	0 - 65535	P
2055	6th PMC encoder resolution (single-phase)	The motor single-frequency pulse number.	0	0 - 65535	P
2056	7th PMC encoder resolution (single-phase)	The motor single-frequency pulse number.	0	0 - 65535	P
2057	8th PMC encoder resolution (single-phase)	The motor single-frequency pulse number.	0	0 - 65535	P
2058	9th PMC encoder resolution (single-phase)	The motor single-frequency pulse number.	0	0 - 65535	P
2059	10th PMC encoder resolution (single-phase)	The motor single-frequency pulse number.	0	0 - 65535	P
621	G00/manual maximum speed	Sets the maximum speed. Unit: mm/min, inch/min, rpm	5000	0 - 60000	R
622	G00/manual ACC/DEC time	Sets the acceleration or deceleration time. Unit: msec	50	0 - 10000	R
623	G00/manual S-curve time	Sets the time constant for S-curve. Unit: msec	5	1 - 2000	R
625	G01 maximum speed	Sets the maximum speed for G01. Unit: mm/min, inch/min, rpm	0	0 - 60000	R
642	Synchronous tolerance	During synchronous control, if the following error between the master and slaves exceeds the value set by this parameter, B645 Excessive synchronous following error occurs. Unit: 0.1 mm	50	0 - 1000	R
643	Feedback following error	If the following error exceeds the setting in any movement, an alarm occurs. This parameter is for setting the tolerance for the servo following command. Unit: 0.001 mm	30000	1 - 60000	R
648	Feedback position check after bootup	When starting, the system checks the error between the command and the feedback from servo. If the error exceeds the setting, the servo cannot switch to Servo on.	20	1 - 6000	R

12.3 Tool magazine parameter (MAGA)

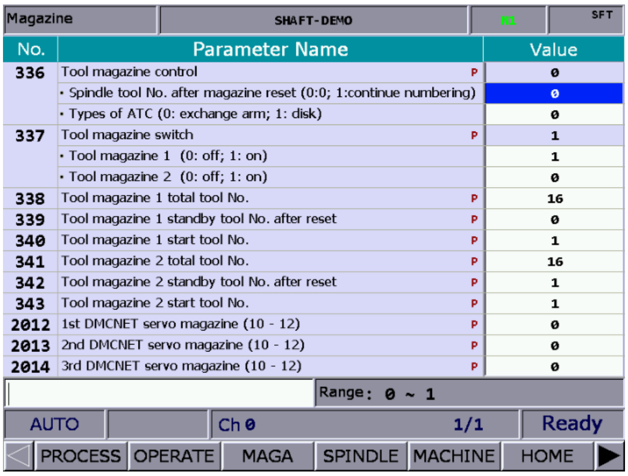
Tool magazine parameters are for enabling magazine-related functions. With these parameters, you can specify the type and number of magazines and determine whether to enable the tool magazine functions. Contact the distributors or equipment suppliers for settings of hardware parameters. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press **MAGA** to enter the tool magazine parameter setting screen.

(3) Press ↑ and ↓ to move the cursor to the specified field, and enter a value within the range specified in the lower right corner of the screen as shown in Figure 12.3.1.

(4) Press **ENTER** to complete the setting.



Magazine SHAFT-DEMO SFT

No.	Parameter Name		Value
336	Tool magazine control	P	0
	• Spindle tool No. after magazine reset (0:0; 1:continue numbering)		0
	• Types of ATC (0: exchange arm; 1: disk)		0
337	Tool magazine switch	P	1
	• Tool magazine 1 (0: off; 1: on)		1
	• Tool magazine 2 (0: off; 1: on)		0
338	Tool magazine 1 total tool No.	P	16
339	Tool magazine 1 standby tool No. after reset	P	0
340	Tool magazine 1 start tool No.	P	1
341	Tool magazine 2 total tool No.	P	16
342	Tool magazine 2 standby tool No. after reset	P	1
343	Tool magazine 2 start tool No.	P	1
2012	1st DMCNET servo magazine (10 - 12)	P	0
2013	2nd DMCNET servo magazine (10 - 12)	P	0
2014	3rd DMCNET servo magazine (10 - 12)	P	0

Range: 0 ~ 1

AUTO Ch 0 1/1 Ready

PROCESS OPERATE MAGA SPINDLE MACHINE HOME

Figure 12.3.1

12.3.1 Tool magazine parameter descriptions

<table>
<tr><th>Para-meter No.</th><th>Item</th><th colspan="3">Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td rowspan="3">336</td><td rowspan="3">Tool magazine control</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="3">0x0800</td><td rowspan="3">0 - 65535</td><td rowspan="3">P</td></tr>
<tr><td>10</td><td>Spindle tool No. after magazine reset
0: 0
1: continue numbering</td><td>0</td></tr>
<tr><td>11</td><td>Types of ATC
0: exchange arm
1: disk</td><td>0 - 1</td></tr>
<tr><td rowspan="3">337</td><td rowspan="3">Tool magazine switch</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="3">1</td><td rowspan="3">0 - 3</td><td rowspan="3">P</td></tr>
<tr><td>0</td><td>Tool magazine 1
0: off (tool magazine data table is not updated instantly)
1: on (tool magazine data table is updated instantly)</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Tool magazine 2
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>338</td><td>Tool magazine 1 total tool No.</td><td colspan="3">Sets the number of tool stations of tool magazine 1 (in response to the mechanical differences of tool magazines, you need to set the number of the placeable tools with this parameter).</td><td>10</td><td>2 - 255</td><td>P</td></tr>
<tr><td>339</td><td>Tool magazine 1 standby tool No. after reset</td><td colspan="3">Sets the standby tool pocket number when the reset function is used for tool magazine 1.</td><td>1</td><td>1 - 100</td><td>P</td></tr>
<tr><td>340</td><td>Tool magazine 1 start tool No.</td><td colspan="3">Sets the tool number of the start tool pocket when the reset function is used for tool magazine 1.</td><td>1</td><td>1 - 100</td><td>P</td></tr>
<tr><td>341</td><td>Tool magazine 2 total tool No.</td><td colspan="3">Sets the number of tool stations of tool magazine 2 (in response to the mechanical differences of tool magazines, you need to set the number of the placeable tools with this parameter).</td><td>10</td><td>2 - 255</td><td>P</td></tr>
<tr><td>342</td><td>Tool magazine 2 standby tool No. after reset</td><td colspan="3">Sets the standby tool pocket number when the reset function is used for tool magazine 2.</td><td>1</td><td>1 - 100</td><td>P</td></tr>
<tr><td>343</td><td>Tool magazine 2 start tool No.</td><td colspan="3">Sets the tool number of the start tool pocket when the reset function is used for tool magazine 2.</td><td>1</td><td>1 - 100</td><td>P</td></tr>
<tr><td>2012</td><td>1st DMCNET servo magazine</td><td colspan="3">Supported station numbers are 10 to 12.</td><td>0</td><td>0 - 65535</td><td>P</td></tr>
<tr><td>2013</td><td>2nd DMCNET servo magazine</td><td colspan="3">Supported station numbers are 10 to 12.</td><td>0</td><td>0 - 65535</td><td>P</td></tr>
<tr><td>2014</td><td>3rd DMCNET servo magazine</td><td colspan="3">Supported station numbers are 10 to 12.</td><td>0</td><td>0 - 65535</td><td>P</td></tr>
</table>

12

12.4 Spindle parameter (SPINDLE)

Spindle parameters are for setting various spindle functions, such as spindle gain, spindle maximum speed, and spindle positioning tolerance. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press **SPINDLE** to enter the spindle parameter setting screen.

(3) Press ↑ and ↓ to move the cursor to the specified field, and enter a value within the range specified in the lower right corner of the screen as shown in Figure 12.4.1.

(4) Press **ENTER** to complete the setting.

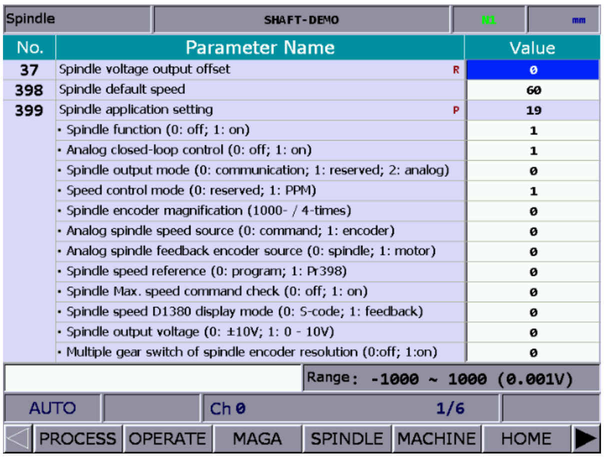


Figure 12.4.1

12.4.1 Spindle parameter descriptions

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>37</td><td>Spindle voltage output offset</td><td>Calibrates the spindle voltage output. When the controller has the spindle stopped (sets the spindle to zero speed), but the spindle is still rotating, set this parameter to adjust the voltage output to have the spindle come to a stop. Unit: 0.001V</td><td>0</td><td>-1000 to +1000</td><td>R</td></tr>
<tr><td>398</td><td>Spindle default speed</td><td>The default spindle speed when power is on.</td><td>0</td><td>0 - 60000</td><td>P</td></tr>
<tr><td>399</td><td>Spindle application setting</td><td>
<table>
<tr><th>Bit</th><th>Function</th><th>Range</th></tr>
<tr><td>0</td><td>Spindle function
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Closed-loop control flag
0: off
1: on (feedback encoder is required)</td><td>0 - 1</td></tr>
<tr><td>2 - 3</td><td>Spindle output mode
0: DMCNET (servo spindle)
1: reserved
2: EDAC (analog output)</td><td>0 - 2</td></tr>
<tr><td>4</td><td>Speed control mode
1: PUU</td><td>1</td></tr>
<tr><td>5</td><td>Spindle encoder magnification
0: 1000 times
1: 4 times</td><td>0 - 1</td></tr>
<tr><td>7</td><td>Analog spindle speed source
0: command
1: encoder</td><td>0 - 1</td></tr>
<tr><td>8</td><td>Analog spindle feedback encoder source
0: spindle
1: motor</td><td>0 - 1</td></tr>
<tr><td>9</td><td>Spindle speed reference
0: program
1: Pr398</td><td>0 - 1</td></tr>
<tr><td>11</td><td>Spindle Max. speed command check
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>12</td><td>Spindle speed D1380 display mode
0: command speed
1: actual speed</td><td>0 - 1</td></tr>
<tr><td>13</td><td>Spindle output voltage (only effective in open-loop control)
0: -10V to +10V
1: 0V to +10V</td><td>0 - 1</td></tr>
<tr><td>14</td><td>Multi-stage spindle feedback encoder switch
0: off
1: on</td><td>0 - 1</td></tr>
</table>
</td><td>0</td><td>0 - 0xFFFF</td><td>P</td></tr>
<tr><td>401</td><td>Spindle encoder port No.</td><td>Sets the feedback channel of spindle encoder.</td><td>8</td><td>0 - 8</td><td>P</td></tr>
<tr><td>402</td><td>Spindle encoder resolution</td><td>Sets the encoder resolution (single-phase).
Unit: pulse/rev</td><td>1280</td><td>2 - 60000</td><td>P</td></tr>
</table>

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>403</td><td>Spindle integral gain</td><td>Sets the speed integral gain. The greater the value, the faster the response. But if the gain is too large, it may cause machine jitter.
Unit: 0.001 rad/s</td><td>50</td><td>1 - 1000</td><td>P</td></tr>
<tr><td>404</td><td>Spindle high positioning speed</td><td>Sets the maximum speed for spindle positioning. If the current speed exceeds the setting of Pr404, the system decreases the spindle speed to the speed set in Pr404 and then performs spindle positioning. In addition, if the spindle speed is lower than the setting of Pr404, it refers to Pr420 Spindle low positioning speed.
Unit: rpm</td><td>100</td><td>1 - 20000</td><td>R</td></tr>
<tr><td>405</td><td>Spindle positioning offset</td><td>Sets the Z-phase offset for spindle positioning.
Unit: 0.01 degree</td><td>0</td><td>0 - 36000</td><td>R</td></tr>
<tr><td>406</td><td>Spindle target speed tolerance</td><td>Sets the allowable tolerance between the target speed and actual speed of the spindle.
Unit: rpm</td><td>10</td><td>0 - 300</td><td>P</td></tr>
<tr><td>407</td><td>Spindle positioning tolerance</td><td>Sets the spindle positioning tolerance. If the actual positioning error exceeds the setting, the spindle positioning fails.
<table>
<tr><th>Positioning error</th><th>M2258</th></tr>
<tr><td>> Pr407</td><td>0</td></tr>
<tr><td>< Pr407</td><td>1</td></tr>
</table>
Unit: 0.01 degree</td><td>100</td><td>0 - 36000</td><td>P</td></tr>
<tr><td>408</td><td>Spindle zero speed tolerance</td><td>If the spindle speed is lower or within the setting, the zero speed signal is on.
(NC > MLC M2257)</td><td>5</td><td>0 - 1000</td><td>P</td></tr>
<tr><td>409</td><td>Spindle maximum speed</td><td>Sets the maximum speed for spindle. Unit: rpm</td><td>20000</td><td>0 - 600000</td><td>P</td></tr>
<tr><td>410</td><td>Spindle minimum speed</td><td>Sets the minimum speed for spindle. Unit: rpm</td><td>10</td><td>0 - 10000</td><td>P</td></tr>
<tr><td>411</td><td>Spindle ACC/DEC time constant</td><td>Sets the acceleration and deceleration time for the spindle. The greater the value, the longer the acceleration and deceleration time.
Unit: msec</td><td>20</td><td>1 - 20000</td><td>R</td></tr>
<tr><td>412</td><td>Spindle S-curve time constant</td><td>Sets the spindle S-curve time constant.
Unit: msec</td><td>10</td><td>1 - 2000</td><td>R</td></tr>
<tr><td>413</td><td>Spindle 2nd Kpp gain</td><td>When the spindle is under closed-loop voltage control, switch M1127 to instantly change the spindle Kpp gain.
<table>
<tr><th>M1127</th><th>Spindle Kpp gain</th></tr>
<tr><td>0</td><td>Pr419</td></tr>
<tr><td>1</td><td>Pr413</td></tr>
</table>
When the spindle performs positioning at low speed, it requires higher gain value while high gain value may lead to vibration during high-speed rotation. Use D1380 Spindle actual speed and this parameter to instantly change the spindle Kpp gain value.</td><td>0</td><td>1 - 1000</td><td>R</td></tr>
<tr><td>416</td><td>Tapping ACC/DEC time constant</td><td>Sets the acceleration and deceleration time of the spindle when it performs tapping.
Unit: msec</td><td>2000</td><td>1 - 20000</td><td>R</td></tr>
</table>

Para-meter No.	Item	Description	Default value	Setting range	Note
417	Tapping S-curve time constant	Sets the S-curve time constant of the spindle when it performs tapping. Unit: msec	100	1 - 2000	R
418	Spindle feedforward gain	The greater the gain, the less the following error. However, if the value is set too high, it may make the control command less smooth and cause machinery vibration.	0	0 - 200	R
419	Spindle Kpp gain	When the spindle is under closed-loop voltage control, use this parameter to adjust the spindle position loop bandwidth. The greater the gain, the more precise the positioning. However, if the value is set too high, it may cause vibration. The parameter value is relevant to the bandwidth of the inverter. It is recommended that you start from a lower gain and then gradually increase the value.	0	0 - 1000	R
420	Spindle low positioning speed	If the current spindle speed is slower than the setting of Pr404 or is zero, when the system performs spindle positioning, it refers to the speed set in this parameter. Unit: rpm	100	1 - 20000	R
421	Tapping retraction speed ratio	Tapping retraction setting. The tapping retraction speed is F multiply by the value of Pr421. Unit: 0.1 times	10	10 - 50	R
422	Gear ratio numerator 1	Sets the numerator of the first set of spindle gear ratio.	1	0 - 60000	P
423	Gear ratio denominator 1	Sets the denominator of the first set of spindle gear ratio.	1	0 - 60000	P
424	Gear ratio numerator 2	Sets the numerator of the second set of spindle gear ratio.	1	0 - 60000	P
425	Gear ratio denominator 2	Sets the denominator of the second set of spindle gear ratio.	1	0 - 60000	P
426	Gear ratio numerator 3	Sets the numerator of the third set of spindle gear ratio.	1	0 - 60000	P
427	Gear ratio denominator 3	Sets the denominator of the third set of spindle gear ratio.	1	0 - 60000	P
428	Gear ratio numerator 4	Sets the numerator of the fourth set of spindle gear ratio.	1	0 - 60000	P
429	Gear ratio denominator 4	Sets the denominator of the fourth set of spindle gear ratio.	1	0 - 60000	P
437	Spindle auxiliary function	Bit: 0 - 1 Function: Proximity switch positioning turn: when using the positioning function of proximity switch, use this parameter to set the number of rotations of the spindle during Z-phase searching. Range: 0 - 3	0	0 - 3	P
438	2nd spindle default speed	The default spindle speed when power is on.	0	0 - 60000	P

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>439</td><td>2nd spindle application setting</td><td>
<table>
<tr><th>Bit</th><th>Function</th><th>Range</th></tr>
<tr><td>0</td><td>Spindle function
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Closed-loop control flag
0: off
1: on (feedback encoder is required)</td><td>0 - 1</td></tr>
<tr><td>2 - 3</td><td>Spindle output mode
0: DMCNET (servo spindle)
1: reserved
2: EDAC (analog output)</td><td>0 - 2</td></tr>
<tr><td>4</td><td>Speed control mode
1: PUU</td><td>1</td></tr>
<tr><td>5</td><td>Spindle encoder magnification
0: 1000 times
1: 4 times</td><td>0 - 1</td></tr>
<tr><td>7</td><td>Analog spindle speed source
0: command
1: encoder</td><td>0 - 1</td></tr>
<tr><td>8</td><td>Analog spindle feedback encoder source
0: spindle
1: motor</td><td>0 - 1</td></tr>
<tr><td>9</td><td>Spindle speed reference
0: program
1: Pr398</td><td>0 - 1</td></tr>
<tr><td>11</td><td>Spindle Max. speed command check
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>12</td><td>Spindle speed D1380 display mode
0: S-code
1: feedback</td><td>0 - 1</td></tr>
</table>
</td><td>0</td><td>0 - 0xFFFF</td><td>P</td></tr>
<tr><td>441</td><td>2nd spindle encoder port No.</td><td>Sets the feedback channel of spindle encoder.</td><td>8</td><td>0 - 8</td><td>P</td></tr>
<tr><td>442</td><td>2nd spindle encoder resolution</td><td>Sets the encoder resolution (single-phase).
Unit: pulse/rev</td><td>1280</td><td>2 - 60000</td><td>P</td></tr>
<tr><td>443</td><td>2nd spindle integral gain</td><td>Sets the speed integral gain. The greater the value, the faster the response. But if the gain is too large, it may cause machine jitter.
Unit: 0.001 rad/s</td><td>50</td><td>1 - 1000</td><td>P</td></tr>
<tr><td>444</td><td>2nd spindle high positioning speed</td><td>Sets the maximum speed for spindle positioning. If the current speed exceeds the setting of Pr444, the system decreases the spindle speed to match the setting of Pr444 and then performs spindle positioning.
In addition, if the spindle speed is lower than the setting of Pr444, it refers to Pr460 Spindle low positioning speed.
Unit: rpm</td><td>100</td><td>1 - 20000</td><td>P</td></tr>
<tr><td>445</td><td>2nd spindle positioning offset</td><td>Sets the Z-phase offset for spindle positioning.
Unit: 0.01 degree</td><td>0</td><td>0 - 36000</td><td>R</td></tr>
</table>

<table>
<tr><th>Para-
mater
No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>446</td><td>2nd spindle target speed tolerance</td><td>Sets the allowable tolerance between the target speed and actual speed of the spindle.
Unit: rpm</td><td>10</td><td>0 - 300</td><td>P</td></tr>
<tr><td>447</td><td>2nd spindle positioning tolerance</td><td>Sets the spindle positioning tolerance. If the actual positioning error exceeds the setting, the spindle positioning fails.<table><tr><th>Positionoing error</th><th>M2258</th></tr><tr><td>> Pr447</td><td>0</td></tr><tr><td>< Pr447</td><td>1</td></tr></table>Unit: 0.01 degree</td><td>100</td><td>0 - 36000</td><td>P</td></tr>
<tr><td>448</td><td>2nd spindle zero speed tolerance</td><td>If the spindle speed is within the tolerance range, the zero speed signal is on.
(NC > MLC M2257)</td><td>5</td><td>0 - 1000</td><td>P</td></tr>
<tr><td>449</td><td>2nd spindle maximum speed</td><td>Sets the maximum speed for spindle. Unit: rpm</td><td>20000</td><td>0 - 600000</td><td>P</td></tr>
<tr><td>450</td><td>2nd spindle minimum speed</td><td>Sets the minimum speed for spindle. Unit: rpm</td><td>10</td><td>0 - 10000</td><td>P</td></tr>
<tr><td>451</td><td>2nd spindle ACC/DEC time constant</td><td>Sets the acceleration and deceleration time for the spindle. The greater the value, the longer the acceleration and deceleration time.
Unit: msec</td><td>20</td><td>1 - 20000</td><td>R</td></tr>
<tr><td>452</td><td>2nd spindle S-curve time constant</td><td>Sets the S-curve time constant of the spindle.
Unit: msec</td><td>10</td><td>1 - 2000</td><td>R</td></tr>
<tr><td>453</td><td>2nd spindle 2nd Kpp gain</td><td>When the spindle is under closed-loop voltage control, switch M1127 to instantly change the spindle Kpp gain.<table><tr><th>M1127</th><th>Spindle Kpp gain</th></tr><tr><td>0</td><td>Pr459</td></tr><tr><td>1</td><td>Pr453</td></tr></table>Low-speed positioning of the spindle requires higher gain value while high-speed positioning with high gain value may lead to vibration. Use D1380 Spindle actual speed and this parameter to instantly change the spindle Kpp gain value.</td><td>0</td><td>1 - 1000</td><td>R</td></tr>
<tr><td>456</td><td>2nd spindle tapping ACC/DEC time constant</td><td>Sets the acceleration and deceleration time of the spindle when it performs tapping.
Unit: msec</td><td>2000</td><td>1 - 20000</td><td>R</td></tr>
<tr><td>457</td><td>2nd spindle tapping S-curve time constant</td><td>Sets the S-curve time constant of the spindle when it performs tapping. Unit: msec</td><td>100</td><td>1 - 2000</td><td>R</td></tr>
<tr><td>458</td><td>2nd spindle feedforward gain</td><td>The greater the gain, the less the following error. However, if the value is set too high, it may make the control command less smooth and cause machinery vibration.</td><td>0</td><td>0 - 200</td><td>R</td></tr>
<tr><td>459</td><td>2nd spindle Kpp gain</td><td>When the spindle is under closed-loop voltage control, use this parameter to adjust the spindle position loop bandwidth. The greater the gain, the more precise the positioning. However, if the value is set too high, it may cause vibration. The parameter value is relevant to the bandwidth of the inverter. It is recommended that you start from a lower gain and then gradually increase the value.</td><td>0</td><td>0 - 1000</td><td>R</td></tr>
</table>

Parameter No.	Item	Description	Default value	Setting range	N
460	2nd spindle low positioning speed	If the current spindle speed is slower than the setting of Pr404 or is zero, when the system performs spindle positioning, it refers to the speed set by this parameter. Unit: rpm	100	1 - 20000	P
461	2nd spindle tapping retraction speed ratio	Tapping retraction setting. The tapping retraction speed is F multiply by the value of Pr421. Unit: 0.1 times	10	10 - 50	R
462	2nd spindle gear ratio numerator 1	Sets the numerator of the first set of the 2nd spindle gear ratio.	1	0 - 60000	P
463	2nd spindle gear ratio denominator 1	Sets the denominator of the first set of the 2nd spindle gear ratio.	1	0 - 60000	P
464	2nd spindle gear ratio numerator 2	Sets the numerator of the second set of the 2nd spindle gear ratio.	1	0 - 60000	P
465	2nd spindle gear ratio denominator 2	Sets the denominator of the second set of the 2nd spindle gear ratio.	1	0 - 60000	P
466	2nd spindle gear ratio numerator 3	Sets the numerator of the third set of the 2nd spindle gear ratio.	1	0 - 60000	P
467	2nd spindle gear ratio denominator 3	Sets the denominator of the third set of the 2nd spindle gear ratio.	1	0 - 60000	P
468	2nd spindle gear ratio numerator 4	Sets the numerator of the fourth set of the 2nd spindle gear ratio.	1	0 - 60000	P
469	2nd spindle gear ratio denominator 4	Sets the denominator of the fourth set of the 2nd spindle gear ratio.	1	0 - 60000	P

12.5 Machine parameter (MACHINE)

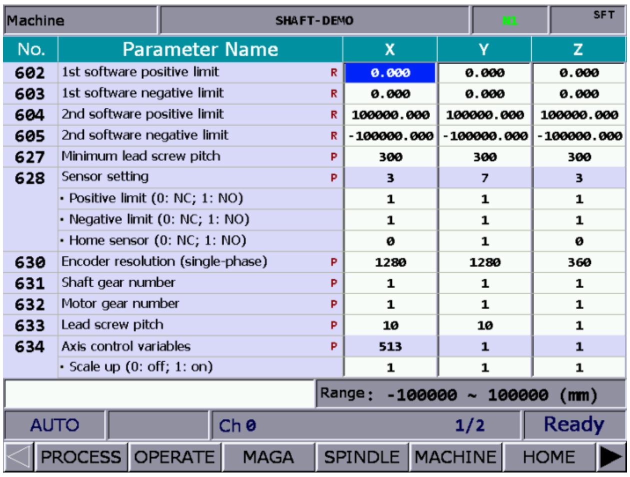
You can set equipment-related parameters, such as software / hardware limit, lead screw pitch, or encoder pulse number, in the machine parameter setting screen. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press **MACHINE** to enter the machine parameter setting screen.

(3) Press ↑ and ↓ to move the cursor to the specified field, and enter a value within the range specified in the lower right corner of the screen as shown in Figure 12.5.1.

(4) Press **ENTER** to complete the setting.



Machine | SHAFT-DEMO | SFT

No.	Parameter Name		X	Y	Z
602	1st software positive limit	R	0.000	0.000	0.000
603	1st software negative limit	R	0.000	0.000	0.000
604	2nd software positive limit	R	100000.000	100000.000	100000.000
605	2nd software negative limit	R	-100000.000	-100000.000	-100000.000
627	Minimum lead screw pitch	P	300	300	300
628	Sensor setting	P	3	7	3
	• Positive limit (0: NC; 1: NO)		1	1	1
	• Negative limit (0: NC; 1: NO)		1	1	1
	• Home sensor (0: NC; 1: NO)		0	1	0
630	Encoder resolution (single-phase)	P	1280	1280	360
631	Shaft gear number	P	1	1	1
632	Motor gear number	P	1	1	1
633	Lead screw pitch	P	10	10	1
634	Axis control variables	P	513	1	1
	• Scale up (0: off; 1: on)		1	1	1

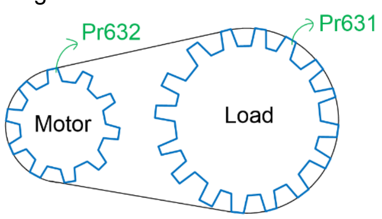
Range: -100000 ~ 100000 (mm)

AUTO | Ch 0 | 1/2 | Ready

PROCESS | OPERATE | MAGA | SPINDLE | MACHINE | HOME

Figure 12.5.1

12.5.1 Machine parameter descriptions

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>602</td><td>1st software positive limit</td><td>Sets the machine coordinates for the 1st software positive limit.
Set Pr602 to 0 to disable this function.
Unit: mm
1. Overtravel will cause software positive limit error.
2. Software limit can be canceled by triggering the special M relay.<table><tr><th>Axis</th><th>Special relays for canceling software limit</th></tr><tr><td>X - W</td><td>M1248 - M1256</td></tr></table></td><td>10^5</td><td>-10^5 to +10^5</td><td>R</td></tr>
<tr><td>603</td><td>1st software negative limit</td><td>Sets the machine coordinates for the 1st software negative limit.
Set Pr603 to 0 to disable this function.
Unit: mm
1. Overtravel will cause software negative limit error.
2. Software limit can be canceled by triggering the special M relay.</td><td>-10^5</td><td>-10^5 to +10^5</td><td>R</td></tr>
<tr><td>604</td><td>2nd software positive limit</td><td>Sets the machine coordinates for the 2nd software positive limit.
Set Pr604 to 0 to disable this function.
Overtravel will cause software positive limit error. Unit: mm</td><td>10^5</td><td>-10^5 to +10^5</td><td>R</td></tr>
<tr><td>605</td><td>2nd software negative limit</td><td>Sets the machine coordinates for the 2nd software negative limit.
Set Pr605 to 0 to disable this function.
Overtravel will cause software negative limit error. Unit: mm</td><td>-10^5</td><td>-10^5 to +10^5</td><td>R</td></tr>
<tr><td>627</td><td>Decimals of lead screw pitch</td><td>Set Pr634 [Bit 9] to 1 to enable the function of decimal places of lead screw pitch. After the function is enabled, the lead screw pitch for the axis is: Pr633 + Pr627 x 0.0001 (mm).
Unit: 0.1 μm</td><td>0</td><td>0 - 9999</td><td>P</td></tr>
<tr><td>628</td><td>Sensor setting</td><td>Sets the input polarity of positive / negative hardware limits and the home sensor.
Sets to 1 as an NO switch.
Sets to 0 as an NC switch.<table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>Positive limit sensor setting</td><td>0 - 1</td></tr><tr><td>1</td><td>Negative limit sensor setting</td><td>0 - 1</td></tr><tr><td>2</td><td>Home sensor setting</td><td>0 - 1</td></tr></table></td><td>0</td><td>0 - 3F</td><td>P</td></tr>
<tr><td>630</td><td>Encoder resolution</td><td>Sets the motor resolution per revolution (single-phase).</td><td>1280</td><td>10 - 50000</td><td>P</td></tr>
<tr><td>631</td><td>Shaft gear number</td><td>Sets the gear number of the counter shaft.

</td><td>1</td><td>1 - 65535</td><td>P</td></tr>
<tr><td>632</td><td>Motor gear number</td><td>Sets the gear number for the motor.</td><td>1</td><td>1 - 65535</td><td>P</td></tr>
</table>

<table>
<tr><th>Para-meter No.</th><th>Item</th><th colspan="3">Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>633</td><td>Lead screw pitch</td><td colspan="3">Sets the corresponding lead screw pitch for the axis. This parameter is only effective for linear axes (i.e., ineffective for rotation axes).
Unit: mm</td><td>10</td><td>2 - 100</td><td>P</td></tr>
<tr><td rowspan="9">634</td><td rowspan="9">Axial control variables</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="9">5</td><td rowspan="9">0 - 65535</td><td rowspan="9">P</td></tr>
<tr><td>0</td><td>Scale up
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>1 - 3</td><td>Rotation axis feed mode (available for A, B, and C axes but not for X, Y, and Z axes.)
0: the rotation axis rotates to the specified position (degree) through a non-shortest path.
1: the rotation axis rotates to the specified position (degree) through the shortest path.
2: shows the position (degree) of the rotation axis in linear representation.
5: sets the rotation axis as a linear axis.</td><td>0 - 5</td></tr>
<tr><td>5</td><td>Encoder magnification
0: 1000 times
1: 4 times</td><td>0 - 1</td></tr>
<tr><td>7</td><td>Rotation axis unit (available for Pr618, Pr619, Pr620, Pr621, and Pr625)
0: rpm
1: deg/min</td><td>0 - 1</td></tr>
<tr><td>9</td><td>Decimal places of lead screw pitch
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>10</td><td>MPG reverse motion
0: moves the machine coordinates towards the positive direction when you operate the MPG in positive direction.
1: moves the machine coordinates towards the positive direction when you operate the MPG in reverse direction.</td><td>0 - 1</td></tr>
<tr><td>15</td><td>Rotation axis reverse motion (lathe)
0: rotates to the specified angle in the original direction.
1: rotates to the specified angle in the reverse direction.</td><td>0 - 1</td></tr>
</table>

12.6 Homing parameter (HOME)

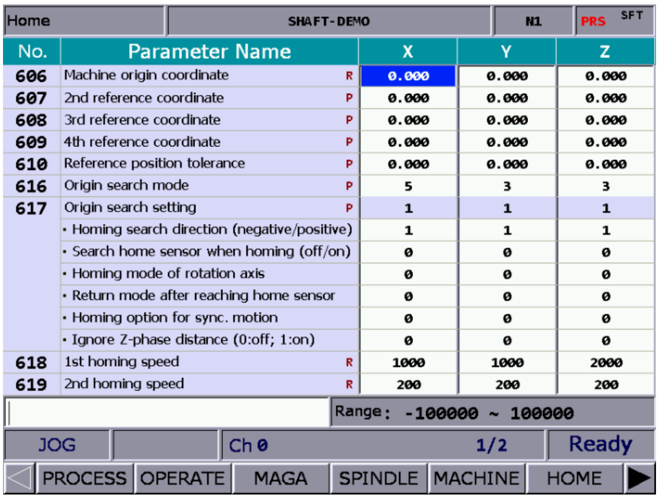
Homing parameters are for setting the origin search mode and the machine coordinates for the machine origin, and the 2nd, 3rd, and 4th reference points. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press **HOME** to enter the homing parameter setting screen.

(3) Press ↑ and ↓ to move the cursor to the specified field, and enter a value within the range specified in the lower right corner of the screen as shown in Figure 12.6.1.

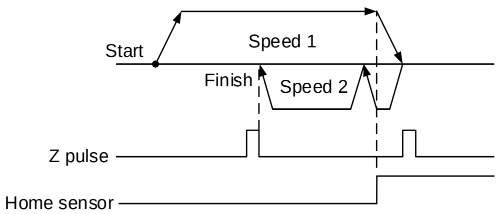
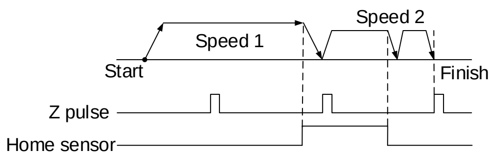
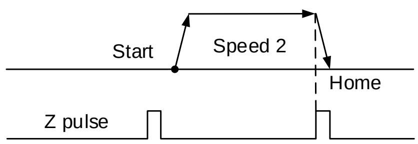
(4) Press **ENTER** to complete the setting.



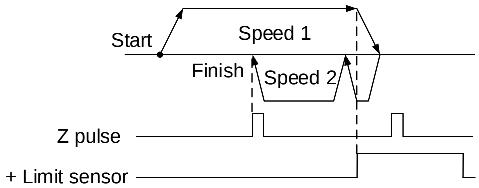
No.	Parameter Name		X	Y	Z
606	Machine origin coordinate	R	0.000	0.000	0.000
607	2nd reference coordinate	P	0.000	0.000	0.000
608	3rd reference coordinate	P	0.000	0.000	0.000
609	4th reference coordinate	P	0.000	0.000	0.000
610	Reference position tolerance	P	0.000	0.000	0.000
616	Origin search mode	P	5	3	3
617	Origin search setting	P	1	1	1
	• Homing search direction (negative/positive)		1	1	1
	• Search home sensor when homing (off/on)		0	0	0
	• Homing mode of rotation axis		0	0	0
	• Return mode after reaching home sensor		0	0	0
	• Homing option for sync. motion		0	0	0
	• Ignore Z-phase distance (0:off; 1:on)		0	0	0
618	1st homing speed	R	1000	1000	2000
619	2nd homing speed	R	200	200	200

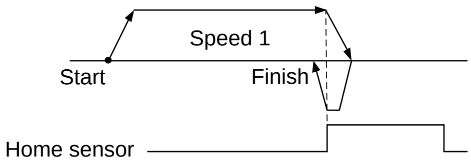
Figure 12.6.1

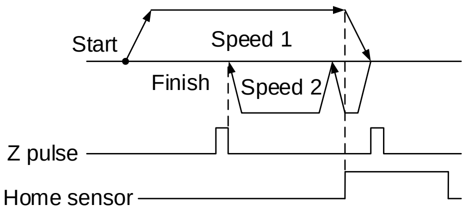
12.6.1 Homing parameter descriptions

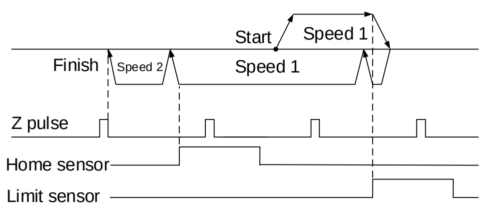
Para-meter No.	Item	Description	Default value	Setting range	Note
606	Machine origin coordinate	After performing homing and finding the Z pulse, the system has the axis offset by the value of Pr606 and refers to the offset position as the machine origin. Unit: CU	0	-10^5 to +10^5	R
607	2nd reference coordinate	Sets the machine coordinates for the 2nd reference point. (Sets the 2nd reference point in the G30 command.) Unit: CU	0	-10^5 to +10^5	P
608	3rd reference coordinate	Sets the machine coordinates for the 3rd reference point. (Sets the 3rd reference point in the G30 command.) Unit: CU	0	-10^5 to +10^5	P
609	4th reference coordinate	Sets the machine coordinates for the 4th reference point. (Sets the 4th reference point in the G30 command.) Unit: CU	0	-10^5 to +10^5	P
610	Reference position tolerance	Sets the position tolerance for the 2nd reference point. For example, when Pr610 = 0.2, it means when the position error between the machine coordinate and the 2nd reference point is within ±0.2 mm, the axis is regarded as reaching the 2nd reference point.	0	-10^8 to +10^8	P
616	Origin search mode	0: off 1: mode 1 When homing, once the motor reaches the home sensor, it reverses and the system regards the first Z pulse as the origin.  2: mode 2 When homing, after the motor reaches the home sensor, it carries on in the same direction to leave the home sensor, and the system regards the first Z pulse as the origin.  3: mode 3 The motor looks for the Z pulse at the 2nd homing speed (Pr619) and the system regards it as the origin. 	1	0 - 24	P

12

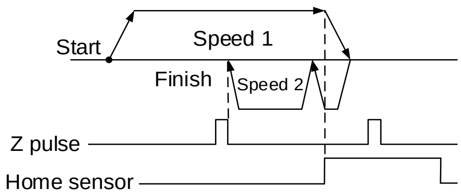
<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>616</td><td>Origin search mode</td><td>
4: mode 4 (OT mode)
When homing, the system regards the positive limit as the home sensor. When the positive limit is triggered, the motor reverses and the system regards the first Z pulse as the origin.


5: mode 5
Absolute motor.
6: mode 6
When homing, once the home sensor is triggered, the motor reverses and decelerates to a stop, and the system regards the stop point as the origin.


7: mode 7 (the function is exclusive to Renishaw’s BiSS C type single-turn absolute motors)
After the servo performs homing for the absolute motor, if the system triggers the special M relays for homing (M1236 - M1241), the system regards the current position as the origin without clearing the machine coordinate.
8: mode 8
There are two possible conditions when homing. One is that the motor first reaches the home sensor and the other is that the motor first reaches the limit sensor.
Condition 1: the motor first reaches the home sensor and reverses to to find the Z pulse.


Condition 2: the motor first reaches the limit sensor and reverses to find the home sensor, and then carries on in the same direction to find the Z pulse.


</td><td>1</td><td>0 - 24</td><td>P</td></tr>
</table>

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>616</td><td>Origin search mode</td><td>24: mode 24
When there is a home sensor on the mechanism and an absolute motor is used, it is recommended to use this mode to perform homing.
The motor first finds the origin in the way as mode 4 does and the system uses the absolute reset function in the DGN screen. After absolute reset, cycle power to the system. The homing mode is automatically switched to mode 5 after power cycling.

</td><td>1</td><td>0 - 24</td><td>P</td></tr>
<tr><td>617</td><td>Origin search setting</td><td><table>
<tr><th>Bit</th><th>Function</th><th>Range</th></tr>
<tr><td>0</td><td>Homing search direction
0: negative
1: positive</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Search home sensor when homing
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>2</td><td>Homing mode of rotation axis
0: single-turn
1: absolute</td><td>0 - 1</td></tr>
<tr><td>3</td><td>Return mode after reaching home sensor
0: return to the machine origin
After finding the Z pulse, the axis offsets by the value set in Pr606. Then, the system regards the current position as the machine origin, completing the homing procedure.
1: return to the Z pulse
After finding the Z pulse, the servo no longer moves, completing the homing procedure.</td><td>0 - 1</td></tr>
<tr><td>4</td><td>Homing option for sync. motion:
0: when the synchronous control function is enabled, the slave axes and the master axis return to the origin synchronously.
1: when the synchronous control function is enabled, each axis can return to the origin individually with the synchronous protection function (Pr642) remains enabled.</td><td>0 - 1</td></tr>
</table></td><td>1</td><td>0 - 31</td><td>P</td></tr>
<tr><td>618</td><td>1st homing speed</td><td>Sets the speed for searching home sensor (Home Dog protector).
Unit: mm/min</td><td>2000</td><td>0 - 10000</td><td>R</td></tr>
<tr><td>619</td><td>2nd homing speed</td><td>Sets the speed for searching the Z pulse.
Unit: mm/min</td><td>200</td><td>0 - 2000</td><td>R</td></tr>
</table>

12

Parameter No.	Item	Description	Default value	Setting range	Note
620	Speed for moving to reference point	Sets the speed for the first homing after system starting with Pr618 and Pr619. After the first homing, the servo refers to the set value of Pr620 for the following homing procedures. Unit: mm/min	10	0 - 20000	R
624	Homing origin protection distance	During homing, when the home sensor is triggered (On), the motor reverses until the signal is off. If the moving distance exceeds the setting of this parameter but the home sensor signal remains on, B636 Home sensor error occurs. Unit: mm	20	1 - 2000	R

12.7 Ethernet setting (ETH.)

You can use Ethernet to connect the system to the PC to enable remote communication. Using the CNCNetwork software and the network setting of NC series controller can manage the online files of multiple NC controllers with one PC, enabling data sharing and file management with the PC, and transmission-along-with-machining (DNC).

The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press ▶ to display the function bar on the second page.

(3) Press **ETH.** to enter the Ethernet setting page.

(4) Press ↑ and ↓ to move the cursor to the specified field, and enter a value within the range specified in the lower right corner of the screen as shown in Figure 12.7.1.

(5) Press **ENTER** to complete the setting.

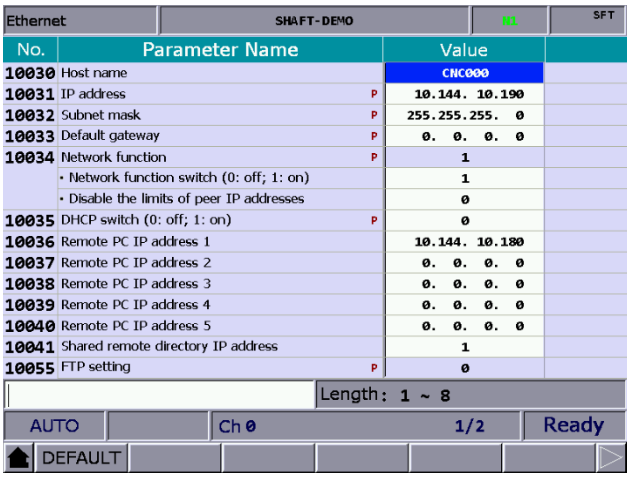


Figure 12.7.1

12.7.1 Ethernet parameter descriptions

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>10030</td><td>Host name</td><td>System’s host name.</td><td>CNC000</td><td>1 - 8</td><td>R</td></tr>
<tr><td>10031</td><td>IP address</td><td>Sets the system IP address.</td><td>0.0.0.0</td><td>0 - 255</td><td>P</td></tr>
<tr><td>10032</td><td>Subnet mask</td><td>Sets the subnet mask of the system.</td><td>0.0.0.0</td><td>0 - 255</td><td>P</td></tr>
<tr><td>10033</td><td>Default gateway</td><td>Sets the system default gateway.</td><td>0.0.0.0</td><td>0 - 255</td><td>P</td></tr>
<tr><td>10034</td><td>Network function</td><td>Sets the network function of the system.
0: off
1: on</td><td>0</td><td>0 - 1</td><td>P</td></tr>
<tr><td>10035</td><td>DHCP switch</td><td>Enable the DHCP function.
0: off
1: on</td><td>0</td><td>0 - 1</td><td>P</td></tr>
<tr><td>10036</td><td>Remote PC
IP address 1</td><td>IP address 1</td><td>0</td><td>255</td><td>P</td></tr>
<tr><td>10037</td><td>Remote PC
IP address 2</td><td>IP address 2</td><td>0</td><td>255</td><td>P</td></tr>
<tr><td>10038</td><td>Remote PC
IP address 3</td><td>IP address 3</td><td>0</td><td>255</td><td>P</td></tr>
<tr><td>10039</td><td>Remote PC
IP address 4</td><td>IP address 4</td><td>0</td><td>255</td><td>P</td></tr>
<tr><td>10040</td><td>Remote PC
IP address 5</td><td>IP address 5</td><td>0</td><td>255</td><td>P</td></tr>
<tr><td>10041</td><td>Shared remote directory IP address</td><td>Specifies an IP address from Pr10036 - Pr10040 for the NETWORK folder under [File manage].
0: do not specify an IP address
1 - 5: specifies the corresponding IP address in Pr10036 - Pr10040</td><td>0</td><td>0 - 5</td><td>P</td></tr>
<tr><td>10055</td><td>FTP setting</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>FTP function switch</td><td>0 - 1</td></tr><tr><td>1</td><td>FTP anonymous login</td><td>0 - 1</td></tr><tr><td>3</td><td>Switch to main file after file upload</td><td>0 - 1</td></tr></table>0: off; 1: on</td><td>0</td><td>0 - 11</td><td>P</td></tr>
<tr><td>10057</td><td>FTP username</td><td>Sets the username with 1 - 6 characters.</td><td>CNCFTP</td><td>-</td><td>P</td></tr>
<tr><td>10058</td><td>FTP password</td><td>Sets the password with 1 - 6 characters.</td><td>123456</td><td>-</td><td>P</td></tr>
</table>

12.8 Compensation parameter (COMP)

During the operation of machine tool, differences in mechanism may lead to motion error, and thus affect the machining result.

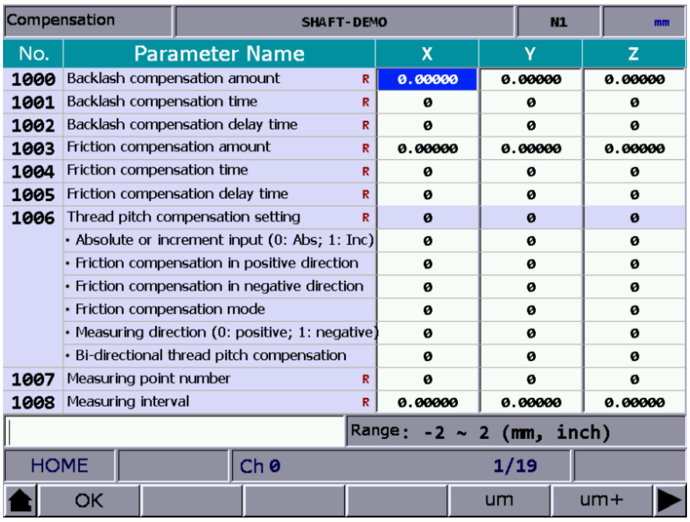
Set the relevant compensation parameters to have the controller compute the appropriate compensation amount according to the machine characteristics. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press ▶ to display the function bar on the second page.

(3) Press **COMP** to enter the compensation parameter setting screen.

(4) Press ⬆ and ⬇ to move the cursor to the specified field, and enter a value within the range specified in the lower right corner of the screen as shown in Figure 12.8.1.



Compensation	SHAFT-DEMO		N1	mm
No.	Parameter Name	X	Y	Z
1000	Backlash compensation amount R	0.00000	0.00000	0.00000
1001	Backlash compensation time R	0	0	0
1002	Backlash compensation delay time R	0	0	0
1003	Friction compensation amount R	0.00000	0.00000	0.00000
1004	Friction compensation time R	0	0	0
1005	Friction compensation delay time R	0	0	0
1006	Thread pitch compensation setting R	0	0	0
	• Absolute or increment input (0: Abs; 1: Inc)	0	0	0
	• Friction compensation in positive direction	0	0	0
	• Friction compensation in negative direction	0	0	0
	• Friction compensation mode	0	0	0
	• Measuring direction (0: positive; 1: negative)	0	0	0
	• Bi-directional thread pitch compensation	0	0	0
1007	Measuring point number R	0	0	0
1008	Measuring interval R	0.00000	0.00000	0.00000

Figure 12.8.1

(5) Press **ENTER** to complete the setting.

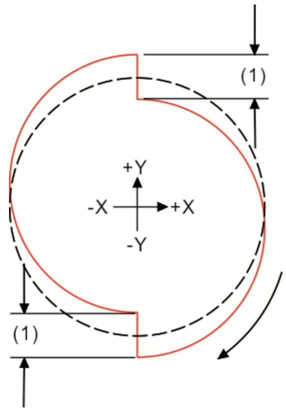
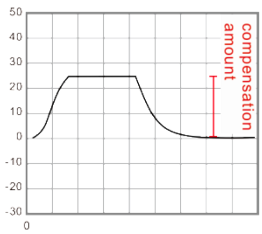
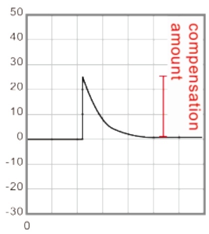
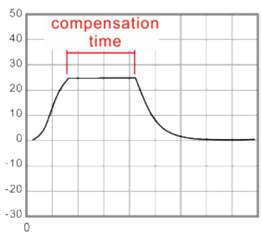
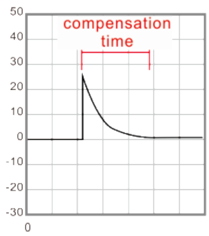
(6) When entering values in the length compensation fields, press **um** to enter absolute values or press **um+** to enter incremental values.

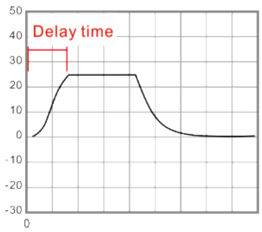
(7) Use the CNCSoft to convert the compensation data measured by the calibration equipment into compensation parameter files. Then, press **IMPORT** on the function bar in the next page to import the data in absolute format. You can also press **IMPORT+** to import the data and add the data to the existing values.

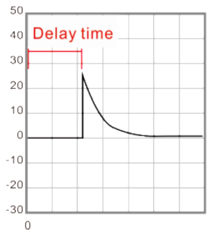
(8) Press **OK** to confirm the update.

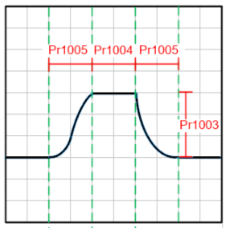
12

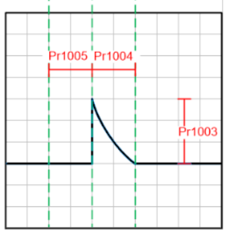
12.8.1 Compensation parameter descriptions

Para-meter No.	Item	Description	Default value	Setting range	Note
1000	Backlash compensation amount	There is backlash in the lead pitch for most of the mechanical systems. Set this parameter with a positive value to compensate for backlash in positive direction, and vice versa. Set this parameter to 0 to disable the backlash compensation function. Unit: mm, inch  (1) Backlash in Y axis	0	-2 to +2	R
1001	Backlash compensation time	Sets the time constant of backlash compensation. When the backlash compensation time is 0, the backlash compensation function is disabled. Unit: 0.1 msec	0	0 - 10000	R
1002	Backlash compensation delay time	Sets the delay time for enabling the backlash compensation function. Unit: 0.1 msec	0	0 - 10000	R
1003	Friction compensation amount	Sets the friction compensation amount. Unit: mm  (Pr1006 [Bit 4] = 0)  (Pr1006 [Bit 4] = 1)	0	0 - 1	R
1004	Friction compensation time	Sets the friction compensation time. Unit: 0.1 msec  (Pr1006 [Bit 4] = 0)  (Pr1006 [Bit 4] = 1)	0	0 - 10000	R

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>1005</td><td>Friction compensation delay time</td><td>Sets the friction compensation delay time.
Unit: 0.1 msec


(Pr1006 [Bit 4] = 0)


(Pr1006 [Bit 4] = 1)</td><td>0</td><td>0 - 10000</td><td>R</td></tr>
<tr><td>1006</td><td>Thread pitch compensation setting</td><td>
<table>
<tr><th>Bit</th><th>Function</th><th>Range</th></tr>
<tr><td>0</td><td>Absolute or incremental input.
0: absolute input (actual value of the measuring point)
1: incremental input (the difference of the current and previous measuring points)</td><td>0 - 1</td></tr>
<tr><td>2</td><td>Friction compensation in positive direction.
Timing: compensates when the machine moves in positive direction
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>3</td><td>Friction compensation in negative direction.
Timing: compensates when the machine moves in reverse direction
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>4</td><td>Friction compensation mode
0: pulse width type




1: exponential type</td><td>0 - 1</td></tr>
<tr><td>15</td><td>Measuring direction (of the start point)
0: positive direction from machine coordinates
1: negative direction from machine coordinates</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Bi-directional thread pitch compensation
0: off
1: on</td><td>0 - 1</td></tr>
</table>
</td><td>0</td><td>0 - 0xFFFF</td><td>R</td></tr>
</table>

12

Para-meter No.	Item	Description	Default value	Setting range	Note
1007	Measuring point number	Sets the number of the measuring points for the lead screw pitch compensation with the maximum as 128. Set this parameter to 0 to disable the compensation function.	0	0 - 128	R
1008	Measuring interval	Sets the interval between each measuring point on the lead screw. Unit: mm	0	0 - 300	R
1009	Measuring offset	Sets the offset between the measuring point and machine origin. For example, when you set this parameter to 0, there will be no offset from the origin; when you set this parameter to 10, there will be an offset of 10 mm from the origin. Note: the direction of the offset should be identical to the direction specified in Pr1006 [Bit 15].	0	-1000 to +1000	R
1010 - 1137	Data 1 - Data 128	Sets the lead screw pitch compensation for the 1st to 128th points. The 1st point and the origin should be the same point. Unit: mm (linear axes), deg (rotation axes)	0	-20 to +20	R
1138 - 1265	Reverse data 1 - Reverse data 128	Sets the lead screw pitch compensation in negative direction for the 1st to 128th points. Enable Pr1006 Bi-directional thread pitch compensation to have this parameter group take effect. Unit: mm (linear axes), deg (rotation axes)	0	-20 to +20	R

12.9 System parameter (SYSTEM)

In the system parameter setting screen, you can change the settings of the system's working environment, such as system date, system time, background color, function bar text color, and label text color. You can set each of these items individually as required. The operation steps are as follows.

(1) Press PAR to enter the PAR screen.

(2) Press ▶ to display the function bar on the second page.

(3) Press **SYSTEM** to enter the system parameter setting screen.

(4) Press ⇧ and ⇩ to move the cursor to the specified field, and enter appropriate values according to the range or format specified in the lower right corner of the screen as shown in Figure 12.9.1.

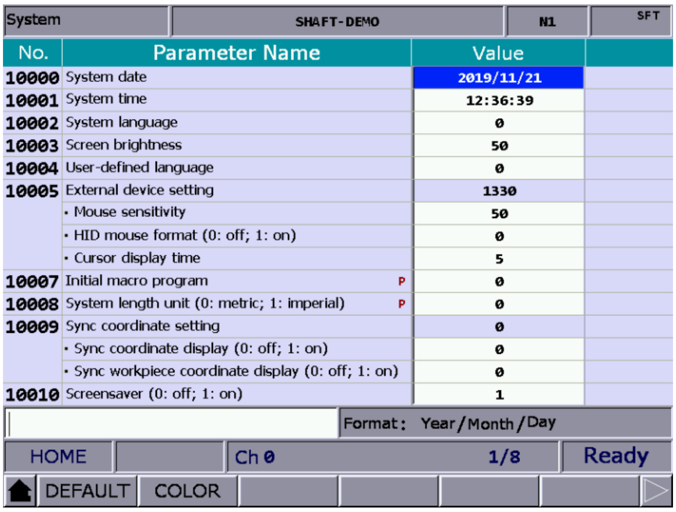


Figure 12.9.1

(5) Press ENTER to complete the setting.

(6) To set parameters for color-related itemss, press **COLOR**, and a color selection box appears for your reference.

(7) To reset the settings, press **DEFAULT**, and a dialog box for confirmation appears.

(8) Enter "Y" and then press ENTER to reset.

12.9.1 System parameter descriptions

<table>
<tr><th>Parameter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>10000</td><td>System date</td><td>Sets the system date (format: yyyy/mm/dd).</td><td>-</td><td>-</td><td>-</td></tr>
<tr><td>10001</td><td>System time</td><td>Sets the system time (format: hh:mm:ss).</td><td>-</td><td>-</td><td>-</td></tr>
<tr><td>10002</td><td>System language</td><td>Sets the system language.
0: English
1: Traditional Chinese
2: Simplified Chinese</td><td>1</td><td>0 - 2</td><td>-</td></tr>
<tr><td>10003</td><td>Screen brightness</td><td>Sets the screen brightness.</td><td>50</td><td>1 - 99</td><td>-</td></tr>
<tr><td>10004</td><td>User-defined language</td><td>Set this parameter to change the language of the software screens. The setting range varies according to the number of languages set by the user.</td><td>0</td><td>0 - 10</td><td>-</td></tr>
<tr><td>10005</td><td>External device setting</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>Mouse sensitivity</td><td>0 - 100</td></tr><tr><td>15</td><td>HID mouse format
0: off
1: on</td><td>0 - 1</td></tr><tr><td>-</td><td>Cursor display time (second)</td><td>1 - 15</td></tr></table></td><td>256</td><td>256 - 36708</td><td>R</td></tr>
<tr><td>10007</td><td>Initial macro program</td><td>Execute the macro program specified by this parameter before pressing CYCLE START.
Note: this program has to be stored in the O_MACRO folder and the naming method is O+Pr10007.</td><td>0</td><td>9000 - 9999</td><td>P</td></tr>
<tr><td>10008</td><td>System length unit</td><td>Sets the unit system for length display on the NC system.
0: metric
1: imperial</td><td>0</td><td>0 - 1</td><td>P</td></tr>
<tr><td>10009</td><td>Sync coordinate setting</td><td>Sets the display of the synchronous coordinates.<table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>Sync coordinate display
0: off
1: on</td><td>0 - 1</td></tr><tr><td>2</td><td>Sync workpiece coordinate display
0: off
1: on</td><td>0 - 1</td></tr></table></td><td>0</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10010</td><td>Screensaver</td><td>Enable the screensaver.
0: off
1: on</td><td>0</td><td>0 - 1</td><td>-</td></tr>
<tr><td>10011</td><td>Screensaver time 1</td><td>Sets the first wait time for the screensaver.</td><td>10</td><td>1 - 60</td><td>-</td></tr>
<tr><td>10012</td><td>Screen brightness 1</td><td>Sets the first level of brightness for the screensaver.</td><td>30</td><td>0 - 99</td><td>-</td></tr>
<tr><td>10013</td><td>Screensaver time 2</td><td>Sets the second wait time for the screensaver.</td><td>30</td><td>1 - 60</td><td>-</td></tr>
<tr><td>10014</td><td>Screen brightness 2</td><td>Sets the second level of brightness for the screensaver.</td><td>10</td><td>0 - 99</td><td>-</td></tr>
</table>

<table>
<tr><th>Para-meter No.</th><th>Item</th><th colspan="3">Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td rowspan="5">10015</td><td rowspan="5">Account setting</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="5">0</td><td rowspan="5">0 - 65535</td><td rowspan="5">P</td></tr>
<tr><td>1</td><td>Account permission activation method
0: by system
1: by external I/O
M2934 = 1 (lock)
M2934 = 0 (unlock)</td><td>0 - 1</td></tr>
<tr><td>2</td><td>Auto open the previous file: after enabling this function, when you insert the USB drive or CF card to the controller, the system automatically opens the last executed file.
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>3</td><td>Auxiliary input window
(This function should be used with the mouse. With the mouse connected to the system, when you click the upper right corner, a list appears; when you click the input window, a keyboard appears.)
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>6</td><td>Machining count display format
0: Word
1: Double word</td><td>0 - 1</td></tr>
<tr><td rowspan="5">10016</td><td rowspan="5">System setting</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="5">4</td><td rowspan="5">0 - 65535</td><td rowspan="5">P</td></tr>
<tr><td>0</td><td>Reset system after EMG release (Reset): sets whether to automatically generate a Reset signal after the emergency stop is released.
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>1</td><td>[SOFT] display after bootup: sets whether to display the SOFT screen as the default screen after system bootup.
0: off
1: switch to SOFT screen after bootup</td><td>0 - 1</td></tr>
<tr><td>2</td><td>Display alarm screen when alarm occurs
0: no alarm display
1: display the alarm</td><td>0 - 1</td></tr>
<tr><td>3</td><td>System parameter auto backup: when this function is enabled, the system automatically makes a backup of the parameter data and stores it to the CF card. Once any of the parameters is modified, the backup data in the CF card is updated as well.
0: off
1: on</td><td>0 - 1</td></tr>
</table>

Parameter No.	Item	Description: Bit	Description: Function	Description: Range	Default value	Setting range	Note
10016	System setting	4	Hidden axis display: if you set not to display an axis in the CONFIG screen, set this parameter to display or hide the coordinate of the corresponding axis. 0: hide the axis coordinate in the CONFIG screen. 1: display the axis coordinate in the CONFIG screen.	0 - 1	4	0 - 65535	P
		5	O macro file protection: once this function is enabled, you can only copy the O macro files to the internal memory rather than copy the files from the internal memory to external devices. 0: off 1: on	0 - 1			
		8	[POS] screen display 0: on 1: off	0 - 1			
		9	[PRG] screen display 0: on 1: off	0 - 1			
		10	[OFS] screen display 0: on 1: off	0 - 1			
		11	[DGN] screen display 0: on 1: off	0 - 1			
		12	[ALM] screen display 0: on 1: off	0 - 1			
		13	[GRA] screen display 0: on 1: off	0 - 1			
		14	[PAR] screen display 0: on 1: off	0 - 1			
		15	[SOFT] screen display 0: on 1: off	0 - 1			
10017	G-code edit setting	0	G-code editing: sets whether to allow G-code editing. 0: off 1: on	0 - 1	1	0 - 65535	-
		1	Macro call file source 0: CF card 1: internal memory	0 - 1			

<table>
<tr><th>Para-meter No.</th><th>Item</th><th colspan="3">Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td rowspan="6">10017</td><td rowspan="6">G-code edit setting</td><td>Bit</td><td>Function</td><td>Range</td><td rowspan="6">1</td><td rowspan="6">0 - 65535</td><td rowspan="6">-</td></tr>
<tr><td>3</td><td>SF speed setting: sets whether you can use the SF SET function to set the cutting feedrate.
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>4</td><td>Program auto reset after editing (Reset): sets whether the cursor will automatically return to the program starting line after file editing.
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>5</td><td>Subprogram call file source
0: CF card
1: internal memory</td><td>0 - 1</td></tr>
<tr><td>6</td><td>Subprogram file name display
0: on
1: off</td><td>0 - 1</td></tr>
<tr><td>7</td><td>.txt file support
0: off
1: on</td><td>0 - 1</td></tr>
<tr><td>10018</td><td>Background color</td><td colspan="3">Sets the background color.</td><td>LIGHTGRAY</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10019</td><td>Title bar text color</td><td colspan="3">Sets the text color for the title bar.</td><td>BLACK</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10020</td><td>Mode bar text color</td><td colspan="3">Sets the text color for the mode bar.</td><td>DARKBLUE</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10021</td><td>Function bar text color</td><td colspan="3">Sets the text color for the function bar.</td><td>BLACK</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10022</td><td>Label text color</td><td colspan="3">Sets the text color for the labels.</td><td>BLACK</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10023</td><td>Numeric value color</td><td colspan="3">Sets the text color for numeric values.</td><td>BLUE</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10024</td><td>Table gridline color</td><td colspan="3">Sets the color of table gridline.</td><td>BLACK</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10025</td><td>System cursor color</td><td colspan="3">Sets the color of the cursor.</td><td>COLOR_S07</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10026</td><td>System text highlight color</td><td colspan="3">Sets the text highlight color.</td><td>WHITE</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10027</td><td>Software panel cursor color</td><td colspan="3">Sets the cursor color in the software panel.</td><td>YELLOW</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10028</td><td>System alarm color</td><td colspan="3">Sets the color of system alarms.</td><td>RED</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10029</td><td>User-defined alarm color</td><td colspan="3">Sets the color of user-defined alarms.</td><td>BLUE</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10042</td><td>Software panel text highlight color</td><td colspan="3">Sets the text highlight color in the software panel.</td><td>COLOR_S07</td><td>0 - 65535</td><td>-</td></tr>
</table>

Parameter No.	Item	Description			Default value	Setting range	Note
		Bit	Function	Range			
10043	[PAR] group item display	0	[OPERATE] display 0: on; 1: off	0 - 1	0	0 - 65535	P
		1	[MAGA] display 0: on; 1: off	0 - 1			
		2	[SPINDLE] display 0: on; 1: off	0 - 1			
		3	[MACHINE] display 0: on; 1: off	0 - 1			
		4	[HOME] display 0: on; 1: off	0 - 1			
		5	[COMP] display 0: on; 1: off	0 - 1			
		6	[SYSTEM] display 0: on; 1: off	0 - 1			
		7	[MLC] display 0: on; 1: off	0 - 1			
		8	[GRAPHIC] display 0: on; 1: off	0 - 1			
		9	[SERVO] display 0: on; 1: off	0 - 1			
		10	[CONFIG] display 0: on; 1: off	0 - 1			
		11	[SET RIO] display 0: on; 1: off	0 - 1			
10044	Channel 0 - teach setting	Enable the Teach function for the corresponding axis. In JOG or MPG mode, set this parameter to enable the Teach function of the PRG group.			0	0 - 65535	P
		Bit	Function	Range			
		0	X axis teaching 0: off; 1: on	0 - 1			
		1	Y axis teaching 0: off; 1: on	0 - 1			
		2	Z axis teaching 0: off; 1: on	0 - 1			
		3	A axis teaching 0: off; 1: on	0 - 1			
		4	B axis teaching 0: off; 1: on	0 - 1			
		5	C axis teaching 0: off; 1: on	0 - 1			
		6	U axis teaching 0: off; 1: on	0 - 1			
		7	V axis teaching 0: off; 1: on	0 - 1			
		8	W axis teaching 0: off; 1: on	0 - 1			
		15	Teaching G-code format 0: moving axes; 1: all axes	0 - 1			

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>10045</td><td>[PRG[/[OFS]/[DGN] group item display</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>[TUNING] display
0: on; 1: off</td><td>0 - 1</td></tr><tr><td>1</td><td>[TEXT WR] display
0: on; 1: off</td><td>0 - 1</td></tr><tr><td>2</td><td>[LOGO WR] display
0: on; 1: off</td><td>0 - 1</td></tr><tr><td>8</td><td>[MACRO] display
0: on; 1: off</td><td>0 - 1</td></tr><tr><td>15</td><td>[FILE QUEUE] display
0: on; 1: off</td><td>0 - 1</td></tr></table></td><td>0</td><td>0 - 65535</td><td>P</td></tr>
<tr><td>10053</td><td>Barcode setting</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0 - 1</td><td>Barcode file reading
0: off; 1: file scan; 2: element</td><td>0 - 2</td></tr><tr><td>2</td><td>Trigger special M relay after reading barcode</td><td>0 - 1023</td></tr><tr><td>12</td><td>Barcode reading file source
0: CF card
1: internal memory</td><td>0 - 1</td></tr></table></td><td>0</td><td>0 - 8190</td><td>P</td></tr>
<tr><td>10054</td><td>Auto logout time</td><td>Sets the auto logout duration. Unit: min</td><td>0</td><td>0 - 1440</td><td>-</td></tr>
<tr><td>10059</td><td>OFS input mode</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>Tool length
0: Auxiliary; 1: Absolute;
2: Incremental</td><td>0 - 2</td></tr><tr><td>2</td><td>Tool wear
0: Auxiliary; 1: Absolute;
2: Incremental</td><td>0 - 2</td></tr><tr><td>4</td><td>Coordinates
0: Auxiliary; 1: Absolute;
2: Incremetal</td><td>0 - 2</td></tr></table></td><td>0</td><td>0 - 42</td><td>-</td></tr>
<tr><td>10060</td><td>Maximum tool wear for a single cut</td><td>Sets the maximum tool wear for a single cut to avoid machining size error. Unit: 0.001 mm</td><td>0</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>10061</td><td>Barcode setting</td><td>Sets the maximum read length.
0: 64 characters (maximum)
1 - 63: 1 character - 63 characters</td><td>0</td><td>0 - 63</td><td>P</td></tr>
</table>

12.10 MLC setting (MLC)

You can set parameters related to the MLC ladder, such as the displayed device and ladder color, in the MLC setting screen. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press ▶ to display the function bar on the second page.

(3) Press **MLC** to enter the MLC setting screen.

(4) Press ⬆ and ⬇ to move the cursor to the specified field, and enter a value within the range specified in the lower right corner of the screen as shown in Figure 12.10.1.

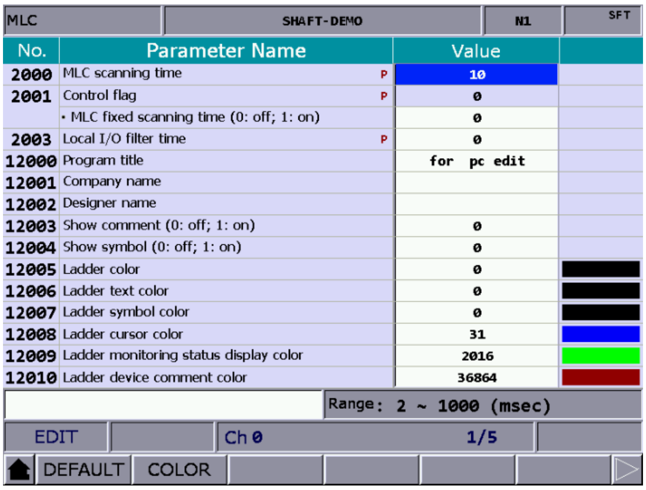


Figure 12.10.1

(5) Press **ENTER** to complete the setting.

(6) To set color-related items, press **COLOR**, and a color selection dialog box appears for your reference.

(7) To reset the settings, press **DEFAULT**, and a dialog box for confirmation appears.

(8) Enter “Y” and then press **ENTER** to reset.

12

12.10.1 MLC parameter descriptions

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>2000</td><td>MLC scanning time</td><td>Sets the MLC scanning time. Unit: msec</td><td>2</td><td>2 - 1000</td><td>P</td></tr>
<tr><td>2001</td><td>Control flag</td><td>MLC fixed scanning time
0: off
1: on</td><td>0</td><td>0 - 1</td><td>P</td></tr>
<tr><td>2003</td><td>Local I/O filter time</td><td>Sets the filter time for local I/O. Unit: msec</td><td>0</td><td>0 - 20</td><td>P</td></tr>
<tr><td>12000</td><td>Program title</td><td>Sets the program title.</td><td>-</td><td>-</td><td>-</td></tr>
<tr><td>12001</td><td>Company name</td><td>Enter the company name.</td><td>0</td><td>-</td><td>-</td></tr>
<tr><td>12002</td><td>Designer name</td><td>Enter the designer name.</td><td>0</td><td>-</td><td>-</td></tr>
<tr><td>12003</td><td>Show comment</td><td>Sets whether to show the comments.
0: off
1: on</td><td>0</td><td>0 - 1</td><td>-</td></tr>
<tr><td>12004</td><td>Show symbol</td><td>Sets whether to show the symbols.
0: off
1: on</td><td>0</td><td>0 - 1</td><td>-</td></tr>
<tr><td>12005</td><td>Ladder color</td><td>Sets the ladder color.</td><td>BLACK</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12006</td><td>Ladder text color.</td><td>Sets the text color for the ladder.</td><td>BLACK</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12007</td><td>Ladder symbol color</td><td>Sets the symbol color for the ladder.</td><td>BLACK</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12008</td><td>Ladder cursor color</td><td>Sets the cursor color for the ladder.</td><td>LIGHT BLUE</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12009</td><td>Ladder monitoring status display color</td><td>Sets the monitoring status display color for the ladder.</td><td>LIGHT GREEN</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12010</td><td>Ladder device comment color</td><td>Sets the comment color of the device for the ladder.</td><td>BROWN</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12011</td><td>Ladder segment comment color</td><td>Sets the comment color of the section for the ladder.</td><td>BROWN</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12012</td><td>Ladder row comment color</td><td>Sets the comment color of the ladder row.</td><td>BROWN</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12013</td><td>Ladder monitoring value color</td><td>Sets the color of the monitoring values for the ladder.</td><td>LIGHT RED</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12014</td><td>NC special device color</td><td>Sets the color of NC special devices.</td><td>COLOR_S2B</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12015</td><td>MLC special device color</td><td>Sets the color of MLC special devices.</td><td>MAGENTA</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>12016</td><td>MLC protection</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>MLC edit protection: when this function is enabled, you can only edit the MLC programs in EDIT mode.
0: off
1: on</td><td>0 - 1</td></tr><tr><td>1</td><td>MLC display
0: on
1: off</td><td>0 - 1</td></tr><tr><td>2</td><td>MLC auto backup: when this function is enabled, the system automatically makes a backup of the MLC data to the CF card.
0: off
1: on</td><td>0 - 1</td></tr></table></td><td>1</td><td>0 - 65535</td><td>P</td></tr>
</table>

Parameter No.	Item	Bit	Function	Range	Default value	Setting range	Note
12016	MLC protection	3	EMG protection for MLC file saving 0: on 1: off	0 - 1	1	0 - 65535	P
		4	MLC file-saving reminder for group switch 0: off 1: on	0 - 1			
12017	MLC setting	0	Immediate take effect after MLC file load: when this function is enabled, MLC files take effect right after being loaded without restarting the system. 0: off 1: on	0 - 1	0	0 - 65535	P
		1	Record system data in special registers 0: off 1: on D1102: Write main file name (O0000 - O9999) D1103, D1104: Total machining time (sec.) D1105, D1106: Single machining time (sec.) D1107: year, month D1108: date, hour D1109: minute, second Note: set hexadecimal format to access D registers and then convert the data into decimal format for use. Example: February 28th, 2020 PM06:45:59 D1107 = 14 02 14 (HEX) = 20 (DEC) (Year = 2000 + 20) 02 (HEX) = 02 (DEC) D1108 = 1C 12 1C (HEX) = 28 (DEC) 12 (HEX) = 18 (DEC) D1109 = 2D 3B 2D (HEX) = 45 (DEC) 3B (HEX) = 59 (DEC)	0 - 1			
		2	Shortcuts for triggering M device 0: off 1: on	0 - 1			
		3	MLC user-defined keypad 0: off 1: on	0 - 1			

12

Para-meter No.	Item	Description			Default value	Setting range	Note
		Bit	Function	Range			
12027	Enable user alarm 0	0	Enable A0 - A15 alarms	0 - 1	0	0 - 65535	P
		1	Enable A16 - A31 alarms	0 - 1			
		2	Enable A32 - A47 alarms	0 - 1			
		3	Enable A48 - A63 alarms	0 - 1			
		4	Enable A46 - A79 alarms	0 - 1			
		5	Enable A80 - A95 alarms	0 - 1			
		6	Enable A96 - A111 alarms	0 - 1			
		7	Enable A112 - A127 alarms	0 - 1			
		8	Enable A128 - A143 alarms	0 - 1			
		9	Enable A144 - A159 alarms	0 - 1			
		10	Enable A160 - A175 alarms	0 - 1			
		11	Enable A176 - A191 alarms	0 - 1			
		12	Enable A192 - A207 alarms	0 - 1			
		13	Enable A208 - A223 alarms	0 - 1			
		14	Enable A224 - A239 alarms	0 - 1			
		15	Enable A240 - A255 alarms	0 - 1			
		Bit	Function	Range			
12028	Enable user alarm 1	0	Enable A256 - A271 alarms	0 - 1	0	0 - 65535	P
		1	Enable A272 - A287 alarms	0 - 1			
		2	Enable A288 - A303 alarms	0 - 1			
		3	Enable A304 - A319 alarms	0 - 1			
		4	Enable A320 - A335 alarms	0 - 1			
		5	Enable A336 - A351 alarms	0 - 1			
		6	Enable A352 - A367 alarms	0 - 1			
		7	Enable A368 - A383 alarms	0 - 1			
		8	Enable A384 - A399 alarms	0 - 1			
		9	Enable A400 - A415 alarms	0 - 1			
		10	Enable A416 - A431 alarms	0 - 1			
		11	Enable A432 - A447 alarms	0 - 1			
		12	Enable A448 - A463 alarms	0 - 1			
		13	Enable A464 - A479 alarms	0 - 1			
		14	Enable A480 - A495 alarms	0 - 1			
		15	Enable A496 - A511 alarms	0 - 1			

12.11 Graph parameter (GRAPHIC)

You can set the graphic display of the motion trajectory in the GRA group with graph parameters. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press ▶ to display the function bar on the second page.

(3) Press **GRAPHIC** to enter the graph parameter setting screen.

(4) Press ⬆ and ⬇ to move the cursor to the specified field, and enter a value within the range specified in the lower right corner of the screen as shown in Figure 12.11.1.

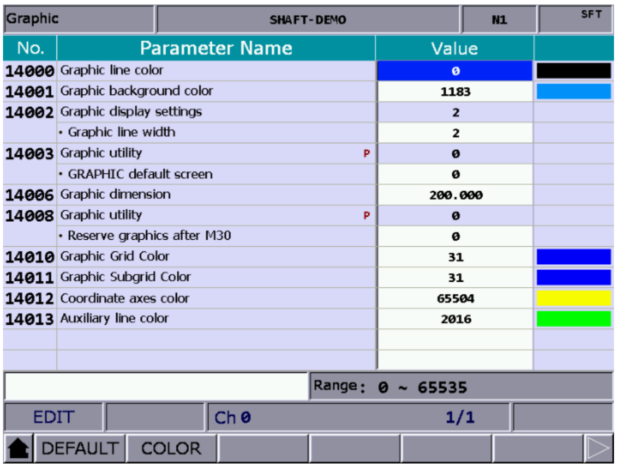


Figure 12.11.1

(5) Press **ENTER** to complete the setting.

(6) To set color-related items, press **COLOR**, and a color selection dialog box appears for your reference.

(7) To reset the settings, press **DEFAULT**, and a dialog box for confirmation appears.

(8) Enter “Y” and then press **ENTER** to reset.

12

12.11.1 Graph parameter descriptions

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>14000</td><td>Line color</td><td>Sets the line color.</td><td>0</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>14001</td><td>Background color</td><td>Sets the background color.</td><td>1183</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>14002</td><td>Graphic display setting</td><td>Sets the graphic display.<table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0 - 3</td><td>Line width</td><td>0 - 4</td></tr></table></td><td>1</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>14003</td><td>Graphic setting</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>Graphic default screen</td><td>0 - 1</td></tr></table></td><td>0</td><td>0 - 65535</td><td>P</td></tr>
<tr><td>14006</td><td>Graphic dimension</td><td>Sets the graphic dimension of the X-Z plane.
Unit: mm</td><td>200</td><td>5 - 100000</td><td>-</td></tr>
<tr><td>14008</td><td>Graphic setting</td><td><table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>1</td><td>Reserve graphics after M30: when this function is enabled, the graphics are automatically reserved after M30 is executed.
0: off
1: on</td><td>0 - 1</td></tr></table></td><td>0</td><td>0 - 65535</td><td>P</td></tr>
<tr><td>14010</td><td>Grid color</td><td>Sets the grid color.</td><td>46516</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>14011</td><td>Subgrid color</td><td>Sets the subgrid color.</td><td>46516</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>14012</td><td>Coordinate axis color</td><td>Sets the color of the coordinate axis.</td><td>65504</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>14013</td><td>Auxiliary line color</td><td>Sets the color of the auxiliary lines.</td><td>2016</td><td>0 - 65535</td><td>-</td></tr>
</table>

12.12 Servo parameter (SERVO)

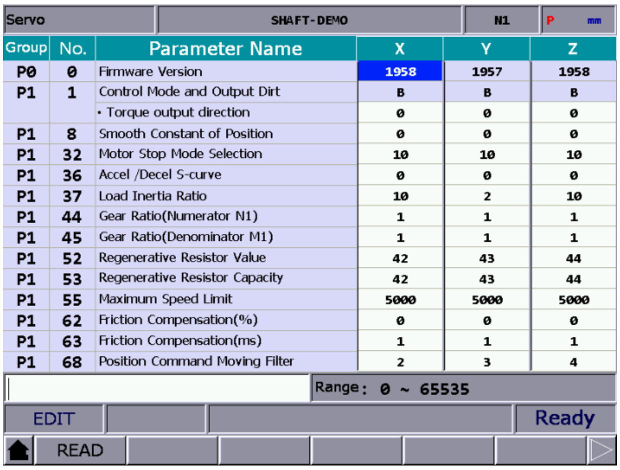
You can set the parameters for the servo drive in the servo parameter setting screen. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press ▶ to display the function bar on the second page.

(3) Press **SERVO** to enter the servo parameter setting screen.

(4) Press ⇧ and ⇩ to move the cursor to the specified field, and enter a value within the range specified in the lower right corner of the screen as shown in Figure 12.12.1.



Group	No.	Parameter Name	X	Y	Z
P0	0	Firmware Version	1958	1957	1958
P1	1	Control Mode and Output Dirt	B	B	B
		• Torque output direction	0	0	0
P1	8	Smooth Constant of Position	0	0	0
P1	32	Motor Stop Mode Selection	10	10	10
P1	36	Accel /Decel S-curve	0	0	0
P1	37	Load Inertia Ratio	10	2	10
P1	44	Gear Ratio(Numerator N1)	1	1	1
P1	45	Gear Ratio(Denominator M1)	1	1	1
P1	52	Regenerative Resistor Value	42	43	44
P1	53	Regenerative Resistor Capacity	42	43	44
P1	55	Maximum Speed Limit	5000	5000	5000
P1	62	Friction Compensation(%)	0	0	0
P1	63	Friction Compensation(ms)	1	1	1
P1	68	Position Command Moving Filter	2	3	4

Figure 12.12.1

(5) Press **ENTER** to complete the setting.

12.12.1 Servo parameter descriptions

<table>
<tr><th>Group</th><th>No.</th><th>Function</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td>P0</td><td>0</td><td>Firmware version</td><td>Displays the firmware version of the servo.</td><td>-</td><td>0</td><td>-</td></tr>
<tr><td>P1</td><td>1</td><td>Input for control mode and control command</td><td>Sets the control mode.<table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>8</td><td>Torque output direction</td><td>0 - 1</td></tr></table></td><td>0</td><td>0x00 - 0x110F (HEX)</td><td>-</td></tr>
<tr><td>P1</td><td>8</td><td>Position command smoothing constant</td><td>The low-pass filter for position command is usually used for eliminating undesired high-frequency response or noise and smoothing and commands. Unit: 10 msec</td><td>0</td><td>0</td><td>Appli-cable to CNC</td></tr>
<tr><td>P1</td><td>36</td><td>S-curve ACC/DEC constant</td><td>The low-pass filter for S-curve is usually used for eliminating undesired high-frequency response or noise and smoothing and commands. Unit: msec</td><td>0</td><td>0</td><td>Appli-cable to CNC</td></tr>
<tr><td>P1</td><td>37</td><td>Load inertia ratio</td><td>Load inertia ratio of servo motor. Unit: 0.1 times</td><td>10</td><td>0 - 2000</td><td>-</td></tr>
<tr><td>P1</td><td>44</td><td>E-gear ratio numerator (N1)</td><td>E-gear ratio numerator. Do not change the setting in the Servo On state. Unit: pulse</td><td>1</td><td>1 to (2^{29}-1)</td><td>Read only</td></tr>
<tr><td>P1</td><td>45</td><td>E-gear ratio denominator (M1)</td><td>E-gear ratio denominator. Do not change the setting in the Servo On state. Unit: pulse</td><td>1</td><td>1 to (2^{31}-1)</td><td>Read only</td></tr>
<tr><td>P1</td><td>55</td><td>Maximum speed limit</td><td>Sets the maximum speed of the servo motor. The default is the rated speed. Unit: rpm</td><td>0</td><td>0 - 65535</td><td>-</td></tr>
<tr><td>P1</td><td>62</td><td>Percentage of friction compensation</td><td>Sets the percentage of friction compensation. Unit: %</td><td>0</td><td>0 - 100</td><td>-</td></tr>
<tr><td>P1</td><td>63</td><td>Constant of friction compensation (ms)</td><td>Sets the smoothing constant of friction compensation. Unit: ms</td><td>4</td><td>4</td><td>Appli-cable to CNC</td></tr>
<tr><td>P1</td><td>68</td><td>Position command - moving filter</td><td>The moving filter smooths the beginning and end of the step command, but it also delays the command. Unit: ms</td><td>4</td><td>0 - 100</td><td>-</td></tr>
<tr><td>P2</td><td>0</td><td>Position control gain</td><td>Increasing the position control gain can enhance the position response and reduce the deviation in position control. If you set the value too high, it may cause vibration and noise. Unit: rad/s</td><td>35</td><td>0 - 2047</td><td>-</td></tr>
<tr><td>P2</td><td>1</td><td>Position control gain rate of change</td><td>Adjusts the rate of change for the position control gain according to the gain switching condition. This parameter is usually used for adjusting the gain of the feeding axis for it to be in accordance with that of the spindle when tapping. Unit: %</td><td>100</td><td>10 - 500</td><td>-</td></tr>
<tr><td>P2</td><td>2</td><td>Position feed forward gain</td><td>If the position control command changes position smoothly, increasing the gain value can reduce the position following errors. If it does not change smoothly, decreasing the gain value can reduce the mechanical vibration during operation. Unit: %</td><td>50</td><td>0 - 100</td><td>-</td></tr>
<tr><td>P2</td><td>3</td><td>Position feed forward gain smoothing constant</td><td>If the position control command changes position smoothly, decreasing the smoothing constant value can reduce the position following errors. If it does not change smoothly, increasing the smoothing constant value can reduce the mechanical vibration during operation. Unit: msec</td><td>5</td><td>2 - 100</td><td>-</td></tr>
</table>

12

Group	No.	Function	Description	Default value	Setting range	Note
P2	4	Speed control gain	Increasing the speed control gain can enhance the speed response. If you set the value too high, it may cause vibration and noise. Unit: rad/s	500	0 - 8191	-
P2	5	Speed control gain rate of change	Adjust the rate of change for the speed control gain according to the gain switching condition. Unit: %	100	10 - 500	-
P2	6	Speed integral compensation	Increasing the value of the integral speed control can enhance the speed response and reduce the deviation in speed control. If you set the value too high, it may cause vibration and noise. Unit: rad/s	100	0 - 1023	-
P2	7	Speed feed forward gain	If the speed control command changes speed smoothly, increasing the gain value can reduce the speed following errors. If it does not change smoothly, decreasing the gain value can reduce the mechanical vibration during operation. Unit: %	0	0 - 100	-
P2	9	DI response filter time	Digital input response filter time. Unit: 2 msec	2	0 - 20	-
P2	23	Notch filter frequency (1)	The first setting for mechanical resonance frequency. Unit: Hz	1000	50 - 1000	-
P2	24	Notch filter attenuation level (1)	The first Notch filter attenuation level. The Notch filter is disabled if this parameter is set to 0. Unit: dB	0	0 - 32	-
P2	25	Resonance suppression low-pass filter	Sets the time constant for the low-pass filter for resonance suppression. The low-pass filter is disabled if this parameter is set to 0. Unit: 0.1 msec	2	0 - 1000	-
P2	26	Anti-interference gain	Increasing this parameter can increase the damping of the speed loop. Setting the value of P2-26 to equal P2-06 is recommended. In Position mode, decrease the value of this parameter to reduce position overshoot. Unit: 0.001	0	0	Appli-cable to CNC
P2	27	Gain switching condition and method selection	When the signal of gain switching is on, the rate of change for the speed control gain is changed to the setting of P2-05.	0	0 - 4 (HEX)	-
P2	28	Gain switching time constant	Controls the switching of smoothing gain. Unit: 10 msec	10	0 - 1000	-
P2	43	Notch filter frequency (2)	The second setting for mechanical resonance frequency. Unit: Hz	1000	50 - 2000	-
P2	44	Notch filter attenuation level (2)	The second Notch filter attenuation level. The Notch filter is disabled if this parameter is set to 0. Unit: dB	0	0 - 32	-
P2	45	Notch filter frequency (3)	The third setting for mechanical resonance frequency. Unit: Hz	1000	50 - 2000	-
P2	46	Notch filter attenuation level (3)	The third Notch filter attenuation level. The Notch filter is disabled if this parameter is set to 0. Unit: dB	0	0 - 32	-
P2	47	Auto resonance suppression mode	0: fixed 1: auto 2: continuous	1	0 - 2	-
P2	49	Speed detection filter and jitter suppression	Sets the filter for speed estimation. Unit: sec	0	0 - 1F	-

Group	No.	Function	Description	Default value	Setting range	Note
P2	53	Position integral compensation	Increasing the position control integral compensation to reduce the position steady-state errors. Unit: rad/s	0	0 - 1023	-
P2	69	Absolute encoder	Sets the operation mode of the motor. Cycle power to the servo to have the setting take effect. 0: incremental type 1: absolute type	0	0 - 1	Cycle power to the servo
P4	0	Fault record (N)	The last abnormal status record.	0	-	Read only
P4	1	Fault record (N-1)	The second to last abnormal status record.	0	-	Read only
P4	2	Fault record (N-2)	The third to last abnormal status record.	0	-	Read only
P4	3	Fault record (N-3)	The fourth to last abnormal status record.	0	-	Read only
P4	4	Fault record (N-4)	The fifth to last abnormal status record.	0	-	Read only
P5	0	Firmware subversion	Displays the firmware subversion of the servo.	0	-	Read only

12.13 Channel setting (CONFIG)

You can enable the axes and define their attributes with this function as shown in Figure 12.13.1. This function is not available in AUTO and MDI modes.

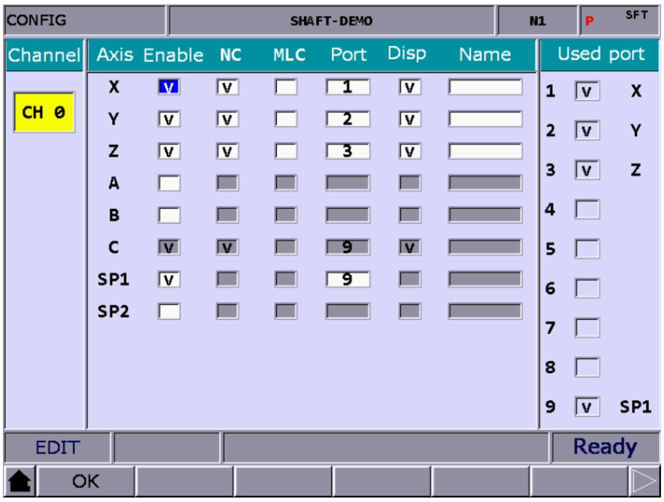


Figure 12.13.1

The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press ▶ to display the function bar on the third page.

(3) Press **CONFIG** to enter the channel setting screen.

(4) The attribute setting fields of an axis that is not enabled are grayed-out. Select the check box of Enable for the axis to set its attributes.

(5) NC / MLC axis selection: set the axis as either an NC or MLC axis.

(6) Port number: port number of the axis should be identical to the station number in the servo system. No. 1 is obligatory while other numbers can be arranged randomly.

(7) After defining all the axes, press **OK**.

(8) Restart the NC system.

Note:

1. To enable an axis, firstly select the check box of the corresponding Enable field. Then, you can set the axis as either an NC axis or MLC axis and set its port number which cannot be identical to other port numbers.
2. To disable an axis, move the cursor to the corresponding Enable field and press **ENTER** to cancel the selection. Then, the axis is disabled.
3. When you change the value of the parameter with a P marked in the Parameter Name field, you have to restart the NC system to have the changed value take effect. When you change the value of the parameter without a P marked in the Parameter Name field, it takes effect immediately without power cycling of the system.

12.14 RIO setting (SET RIO)

The NC system can add the control switches for external devices with the I/O extension modules. You can enable the I/O module in the RIO Setting screen as shown in Figure 12.14.1.

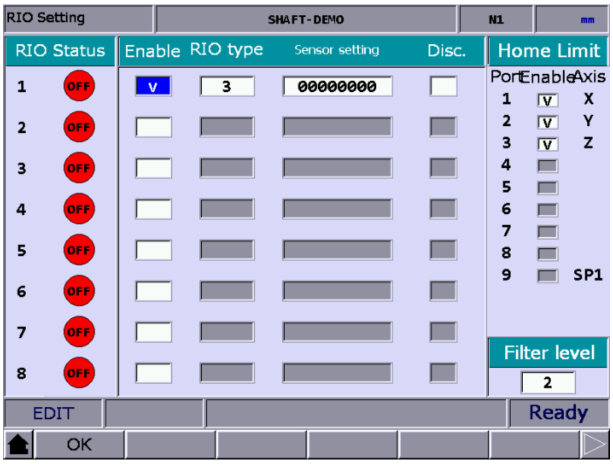


Figure 12.14.1

The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press ▶ to display the function bar on the third page.

(3) Press **SET RIO** to enter the RIO Setting page.

(4) Press ↑ and ↓ to move the cursor to the corresponding Enable field of the specified RIO port, and press **ENTER** to select the check box and enable its corresponding settings.

(5) Press ← and → to move the cursor to the Sensor setting field, press **ENTER**, and an input window appears. After entering the value, press **ENTER** to complete the setting.

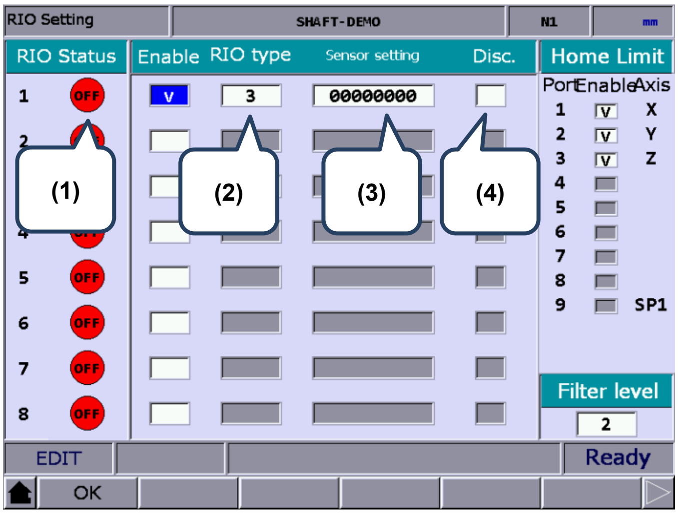
(6) Press ← and → to move the cursor to the Disc. field, and press **ENTER** to select or cancel the selection.

(7) After enabling and setting all the I/O modules, press **OK** to complete the setting.

12

12.14.1 Details of RIO setting

RIO: press **OK** after completing the settings.



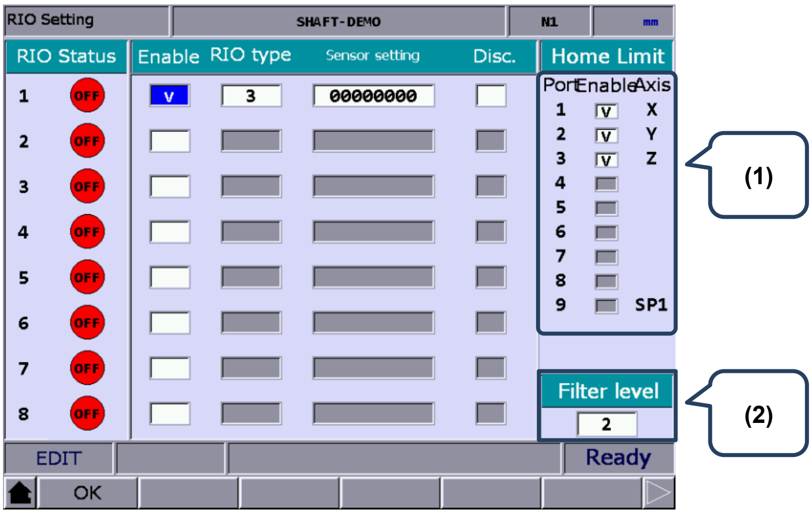
(1) Displays the status of the RIO ports in sequence according to the station numbers.

(2) Select the check box of the corresponding Enable field and set the RIO type to 0 as AD/DA, 1 as DA, 2 as AD, or 3 as DIO.

(3) DI input of the RIO can be set as the positive limit, negative limit, and home sensor of each axis, while only the DIs of Station 0 (the first RIO board) can be set. DIs on other RIO boards cannot be set. You can set 32 points in total from DI0 to DI31.

(4) Select the Disc. Field to have the DO remain its status when it is disconnected from the controller.

Home Limit: press **OK** after completing the setting.



(1) Axis selection: positive limit, negative limit, and home limit DI of X - A axes are input from the AXIS 1~4 connector on the controller (this connector is only available on NC3XX series models). According to the selected axes, each axis takes three DI points from X256, which are positive limit DI, negative limit DI, and home limit DI respectively.

For example, if you select Y and Z axes, the positive limit, negative limit, and home limit for each axis are as shown in the following table.

Signal \ Axis	X axis	Y axis	Z axis
Positive limit	AXIS_P1	X256	X259
Negative limit	AXIS_P2	X257	X260
Home	AXIS_P3	X258	X261

The special M relay code corresponding to the DI signal of each axis does not change regardless of the signal source.

Signal \ Axis	X axis	Y axis	Z axis
Positive limit	M2144	M2148	M2152
Negative limit	M2145	M2149	M2153
Home	M2146	M2150	M2154

(2) Sets the filter level of the DI on the RIO board. The interval between each level is 40 ms and there are 5 levels in total.

Level 0	Level 1	Level 2	Level 3	Level 4	Level 5
200 μs	200 μs	400 μs	600 μs	800 μs	1 ms

12.14.2 Setting DAC module (NC-EIO-DAC04)

The following steps illustrate how to set the DAC (Digital to analog converter) module.

(1) In the RIO Setting screen, enable the 5th port. You can only set the 5th to 8th ports of the DAC module.

(2) Set the RIO type to 1 and D1464 - D1467 correspond to the output points 0 - 3 on the module card.

(3) Rotate the station knob of the DAC module to 4.

(4) Connect the DAC module to the controller in the same way as connecting the RIO.

(5) After complete the above four steps, cycle power to the system. Then, set the value 1024 to D1464, and you can measure 1.25V at the output point 0 on the DAC module (-/+10V correspond to -8191 to +8192).

Refer to the following table for the RIO port numbers and their corresponding MLC special register addresses.

DAC / Port No.	5	6	7	8
Output point 0	D1464	D1472	D1480	D1488
Output point 1	D1465	D1473	D1481	D1489
Output point 2	D1466	D1474	D1482	D1490
Output point 3	D1467	D1475	D1483	D1491

12.15 Search (SEARCH)

Use the Search function to find the specified parameter field by entering the parameter number, which is a faster and easier way to display the specified parameter. The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press ▶ to display the function bar on the third page.

(3) Enter the parameter number to be searched to the input field in the lower left corner of the screen.

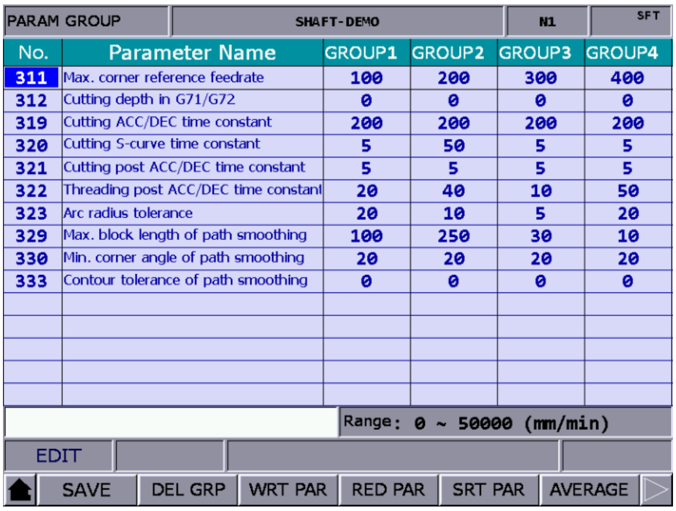
(4) Press **SEARCH** to search for the specified parameter.

Note: in addition to using the Search function, you can also search for the parameter in the PAR screen by entering “S + parameter number” and pressing **ENTER**.

12.16 Parameter group (PAR GRP)

Various types of parameters are available in the NC system. Equipment distributors can use the parameter group function to provide users with the most appropriate combination of parameters according to industrial requirements for machining, which simplifies the complicated parameter adjustments.

With this function, you can use G05P_ to switch the parameter groups during machining as shown in Figure 12.16.1.



No.	Parameter Name	GROUP1	GROUP2	GROUP3	GROUP4
311	Max. corner reference feedrate	100	200	300	400
312	Cutting depth in G71/G72	0	0	0	0
319	Cutting ACC/DEC time constant	200	200	200	200
320	Cutting S-curve time constant	5	50	5	5
321	Cutting post ACC/DEC time constant	5	5	5	5
322	Threading post ACC/DEC time constant	20	40	10	50
323	Arc radius tolerance	20	10	5	20
329	Max. block length of path smoothing	100	250	30	10
330	Min. corner angle of path smoothing	20	20	20	20
333	Contour tolerance of path smoothing	0	0	0	0

Figure 12.16.1

The operation steps are as follows.

(1) Press **PAR** to enter the PAR screen.

(2) Press ▶ to display the function bar on the third page.

(3) Press **PAR GRP** to enter the parameter group setting page.

(4) Enter the specified parameter number in the No. field and press **ENTER** to display the corresponding parameter name.

(5) Press **SRT PAR** to display the parameters by numbers in ascending or descending order.

(6) Press ← and → to move the cursor to the Group field, enter the value, and press **ENTER** to complete setting the parameter group. You can also stop the cursor at the Group field, press **RED PAR**, and a confirmation window appears. Enter “Y” and press **ENTER** to access the current parameter values and write them to the corresponding fields.

(7) To delete a group, press ← and → to move the cursor to any of the fields of the group to be deleted, press **DEL GRP**, and a confirmation window appears. Enter “Y” and press **ENTER** to delete the group.

(8) After enabling multiple groups, press **AVERAGE** and a confirmation window appears. Enter “Y” and press **ENTER**, and the system divides the maximum of the setting parameter by the number of currently enabled groups and defines the quotient as the first term, assigning values to each group field in arithmetic progression with the common difference the same as the first term.

(9) Press **SAVE**, and a confirmation window appears. Enter “Y” and press **ENTER** to save the settings.

(10) After setting the groups, press ← and → to move the cursor to the specified group field, then press **WRT PAR**, and a confirmation window appears. Enter “Y” and press **ENTER** to write the values to corresponding parameters.

Note:

1. The parameter write function overwrites the original values, so ensure the new values are correct before using this function.
2. The parameter group function supports up to 20 parameters and 20 groups.

12.17 Other settings

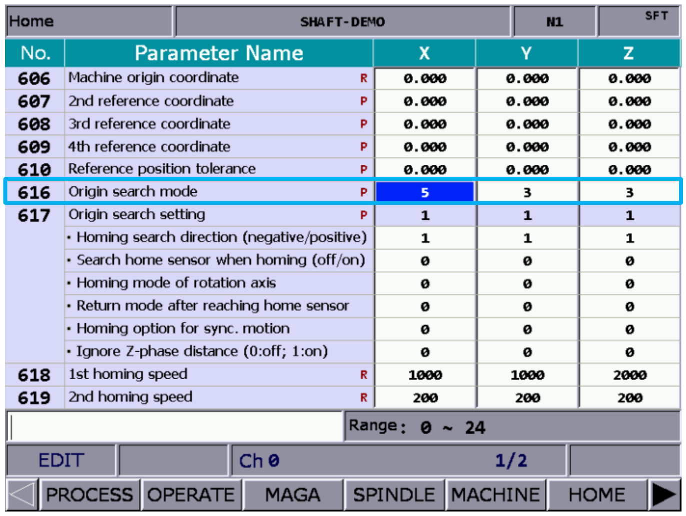
12.17.1 Setting for absolute motor

12

Follow these steps to set the system when using the NC series controller with an absolute motor.

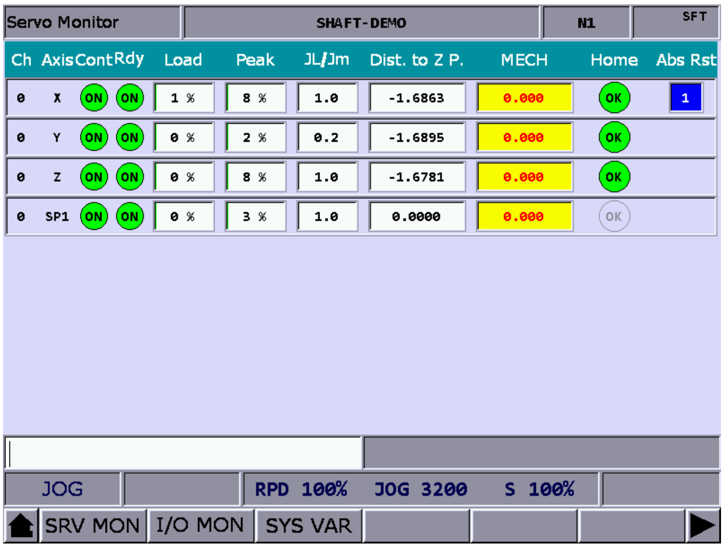
(1) In the homing parameter screen, set Pr616 Origin search mode to 5 (either an incremental or absolute encoder can be used. When you use an absolute motor for the first time, cycle power to the servo and controller after setting the parameter.)

Refer to the following figure.



No.	Parameter Name		X	Y	Z
606	Machine origin coordinate	R	0.000	0.000	0.000
607	2nd reference coordinate	P	0.000	0.000	0.000
608	3rd reference coordinate	P	0.000	0.000	0.000
609	4th reference coordinate	P	0.000	0.000	0.000
610	Reference position tolerance	P	0.000	0.000	0.000
616	Origin search mode	P	5	3	3
617	Origin search setting	P	1	1	1
	• Homing search direction (negative/positive)		1	1	1
	• Search home sensor when homing (off/on)		0	0	0
	• Homing mode of rotation axis		0	0	0
	• Return mode after reaching home sensor		0	0	0
	• Homing option for sync. motion		0	0	0
	• Ignore Z-phase distance (0:off; 1:on)		0	0	0
618	1st homing speed	R	1000	1000	2000
619	2nd homing speed	R	200	200	200

(2) After setting the parameter, to reset the absolute encoder, go to **DGN** > **SYS MON** > **SRV MON** as shown in the following figure.



(3) Set the system to JOG or MPG mode to use the absolute reset function. In JOG or MPG mode, move the axis to the position to be defined as the origin, enter “1” and press **ENTER** to complete the setting. Meanwhile, the Home indicator is on, meaning that this axis has completed homing.

Note: when a servo alarm occurs, the special M relay for absolute reset (Abs Rst) becomes 0, meaning that the absolute origin is lost. The following are the relevant alarms for absolute reset.

AL060: absolute position is lost. Perform absolute reset.

AL061: battery undervoltage. Replace the battery.

AL069: wrong encoder. Ensure an absolute encoder is connected.

12

12.17.2 Setting synchronous motion control

Application description: the A axis (slave axis) is required to follow the Z axis (master axis) in the same direction. Assume that M13 is to enable the synchronous function and M14 is to disable it, the settings are as follows.

1. Set Pr350 to 13.
2. Set Pr351 to 14.
3. Set Pr364 Synchronous control A to 3.

When M13 is executed, the MLC triggers M1088 (Trigger for synchronous control) and M1092 (A slave axis follows the master axis) at the same time. When the system commands the Z axis to move, the A axis moves synchronously. If there is a command to move the A axis when the synchronous function is enabled, an alarm occurs since the slave axis (A axis) cannot receive a motion command from the system. Execute M14 to turn off M1088 to stop the synchronization function. The synchronous control function is effective in AUTO, MDI, JOG, MPG, and HOME modes.

Important:

(1) Once you set an axis as a master axis, you cannot set it as a slave axis.
(2) Once you set an axis as a slave axis, you cannot set it as a master axis.
(3) Multiple slave axes can follow the same master axis.
(4) Pressing **RESET** does not disable the synchronous control function.
(5) The synchronous control function is not available during tapping.

Program:

```
G54X0Y0Z0A0
G90G54G0X10.Y10.Z10.
Z50.
A0
M13
Z0.
Z111.
G4X2.
Z150.
M14
A100.
A51.
M30
```

Relevant parameters:

<table>
<tr><th>Para-meter No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td rowspan="8">350 - 357</td><td rowspan="8">Halt M-code 1 - 8</td><td>Halt M-code 1
0: disabled</td><td rowspan="8">0</td><td rowspan="8">0 - 1000</td><td rowspan="8">P</td></tr>
<tr><td>Halt M-code 2</td></tr>
<tr><td>Halt M-code 3</td></tr>
<tr><td>Halt M-code 4</td></tr>
<tr><td>Halt M-code 5</td></tr>
<tr><td>Halt M-code 6</td></tr>
<tr><td>Halt M-code 7</td></tr>
<tr><td>Halt M-code 8</td></tr>
<tr><td>360</td><td>Synchronization direction control</td><td>Sets the synchronization direction.
Bit 0 - 5: synchronous control of X - C axes
0: same direction
1: different directions<table>
<tr><th>Bit</th><th>Function</th><th>Range</th></tr>
<tr><td>0</td><td>Synchronous direction for X axis</td><td>0 - 1</td></tr>
<tr><td>1</td><td>Synchronous direction for Y axis</td><td>0 - 1</td></tr>
<tr><td>2</td><td>Synchronous direction for Z axis</td><td>0 - 1</td></tr>
<tr><td>3</td><td>Synchronous direction for A axis</td><td>0 - 1</td></tr>
<tr><td>4</td><td>Synchronous direction for B axis</td><td>0 - 1</td></tr>
<tr><td>5</td><td>Synchronous direction for C axis</td><td>0 - 1</td></tr>
</table></td><td>0</td><td>0 - 0x3F</td><td>P</td></tr>
<tr><td>361</td><td>Synchronous control X</td><td>Specifies the master axis when X axis is the slave axis. For example, set this parameter to 2 if desiring to set Y axis as the master axis for synchronous control.
0: disabled; 1- 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>362</td><td>Synchronous control Y</td><td>Specifies the master axis when Y axis is the slave axis.
0: disabled; 1- 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>363</td><td>Synchronous control Z</td><td>Specifies the master axis when Z axis is the slave axis.
0: disabled; 1- 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>364</td><td>Synchronous control A</td><td>Specifies the master axis when A axis is the slave axis.
0: disabled; 1- 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>365</td><td>Synchronous control B</td><td>Specifies the master axis when B axis is the slave axis.
0: disabled; 1- 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>366</td><td>Synchronous control C</td><td>Specifies the master axis when C axis is the slave axis.
0: disabled; 1- 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
</table>

12

Special M relays for enabling synchronous control function:

Function description	Special relay code
Trigger for synchronous control	M1088
X slave axis follows the master axis	M1089
Y slave axis follows the master axis	M1090
Z slave axis follows the master axis	M1091
A slave axis follows the master axis	M1092
B slave axis follows the master axis	M1093
C slave axis follows the master axis	M1094

12.17.3 Command transfer

Application description: transfer the command for Z axis to A axis (transfer axis). Assume that M20 is to enable transfer and M21 is to disable it, the settings are as follows.

1. Set Pr350 to 20.
2. Set Pr351 to 21.
3. Set Pr374 Transfer control A to 3.

After executing M20 to trigger M1098 (Trigger for transfer command) and M1102 (A axis receives command from master axis), the system transfers the command that moves Z axis to A axis (that is, Z axis does not move). If a command that moves the A axis is executed, an alarm occurs since the transfer axis (A axis) cannot receive a motion command. Execute M21 to turn off M1098 to stop the command transfer control function. Enabling (M20) and disabling (M21) this function are only available in AUTO and MDI modes. Execute M21 to end the program. The command transfer function is not available in JOG, MPG, and HOME modes.

Important:

(1) Once you set an axis as a transfer axis, you cannot set it as a master axis.
(2) Once you set an axis as a master axis, you cannot set it as a transfer axis.
(3) Multiple transfer axes can refer to the same master axis.
(4) The transfer function is not available in HOME mode.
(5) Pressing **RESET** does not disable the command transfer function.
(6) When the command is transferred from Z axis to A axis, the tool length compensation function is available.
(7) A cutting cycle command for Z axis can be transferred.

Program:

```
G54X0Y0Z0A0
G90G54G0X10.Y10.Z10.
Z50.
A0
M20   (The controller pre-reads M20 and then enables command transfer control.)
Z0.   (The Z-axis command actually moves the A axis)
Z111.
G4X2.
Z150.
M21   (The controller pre-reads M21 and then disables command transfer control.)
A100.
A51.
M30
```

Relevant parameters:

<table>
<tr><th>Para-
meter
No.</th><th>Item</th><th>Description</th><th>Default value</th><th>Setting range</th><th>Note</th></tr>
<tr><td rowspan="8">350 - 357</td><td rowspan="8">Halt M-code
1 - 8</td><td>Halt M-code 1
0: disabled</td><td rowspan="8">0</td><td rowspan="8">0 - 1000</td><td rowspan="8">P</td></tr>
<tr><td>Halt M-code 2</td></tr>
<tr><td>Halt M-code 3</td></tr>
<tr><td>Halt M-code 4</td></tr>
<tr><td>Halt M-code 5</td></tr>
<tr><td>Halt M-code 6</td></tr>
<tr><td>Halt M-code 7</td></tr>
<tr><td>Halt M-code 8</td></tr>
<tr><td>370</td><td>Transfer control direction</td><td>Sets the transfer control direction.
Bit 0 - 5: transfer direction of X - C axes
0: same direction
1: different directions<table><tr><th>Bit</th><th>Function</th><th>Range</th></tr><tr><td>0</td><td>Transfer direction X</td><td>0 - 1</td></tr><tr><td>1</td><td>Transfer direction Y</td><td>0 - 1</td></tr><tr><td>2</td><td>Transfer direction Z</td><td>0 - 1</td></tr><tr><td>3</td><td>Transfer direction A</td><td>0 - 1</td></tr><tr><td>4</td><td>Transfer direction B</td><td>0 - 1</td></tr><tr><td>5</td><td>Transfer direction C</td><td>0 - 1</td></tr></table></td><td>0</td><td>0 - 0x3F</td><td>P</td></tr>
<tr><td>371</td><td>Transfer control X</td><td>Specifies the X axis as the axis to receive the transfer command. When transfer control is enabled, the command is transferred to have the X axis move while the originally commanded axis does not move. For example, set this parameter to 2 if desiring to transfer the control command from the Y axis.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>372</td><td>Transfer control Y</td><td>Specifies the Y axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the Y axis move while the originally commanded axis does not move.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>373</td><td>Transfer control Z</td><td>Specifies the Z axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the Z axis move while the originally commanded axis does not move.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>374</td><td>Transfer control A</td><td>Specifies the A axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the A axis move while the originally commanded axis does not move.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
<tr><td>375</td><td>Transfer control B</td><td>Specifies the B axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the B axis move while the originally commanded axis does not move.
0: disabled; 1 - 6: X - C</td><td>0</td><td>0 - 6</td><td>P</td></tr>
</table>

Parameter No.	Item	Description	Default value	Setting range	Note
376	Transfer control C	Specifies the C axis as the axis to receive the transfer command. When transfer control function is enabled, the command is transferred to have the C axis move while the originally commanded axis does not move. 0: disabled; 1 - 6: X - C	0	0 - 6	P

Relevant special M relays for transfer function:

Function description	Special relay code
Trigger for transfer command	M1098
X axis receives command from master axis	M1099
Y axis receives command from master axis	M1100
Z axis receives command from master axis	M1101
A axis receives command from master axis	M1102
B axis receives command from master axis	M1103
C axis receives command from master axis	M1104
Transfer function in execution	M2228

(This page is intentionally left blank.)

Software (SOFT) Group 13

The SOFT group is for configuring user-defined screens with the CNCSoft software. This chapter provides the example screens.

With the SOFT group function, you can use ScreenEditor to configure the screens, which can replace the function of the machine operation panel B or add customized extension functions.

Note: bold function names in a box (such as **POS**) mean the keys on machine operation panel A; bold function names (such as **CLR ALL**) mean the function keys of F1 - F6.

13.1 ScreenEditor software

■ ScreenEditor

You can edit the screens of the controller with ScreenEditor which you can enter from the main page of the Delta CNCSoft software as shown in Figure 13.1.1.

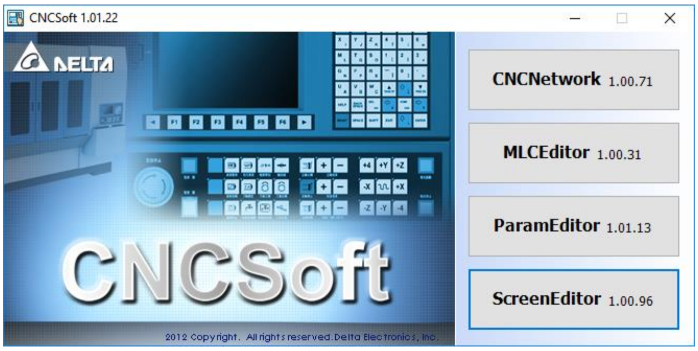


Figure 13.1.1

■ After entering ScreenEditor, you can see the operation interface as shown in Figure 13.1.2.

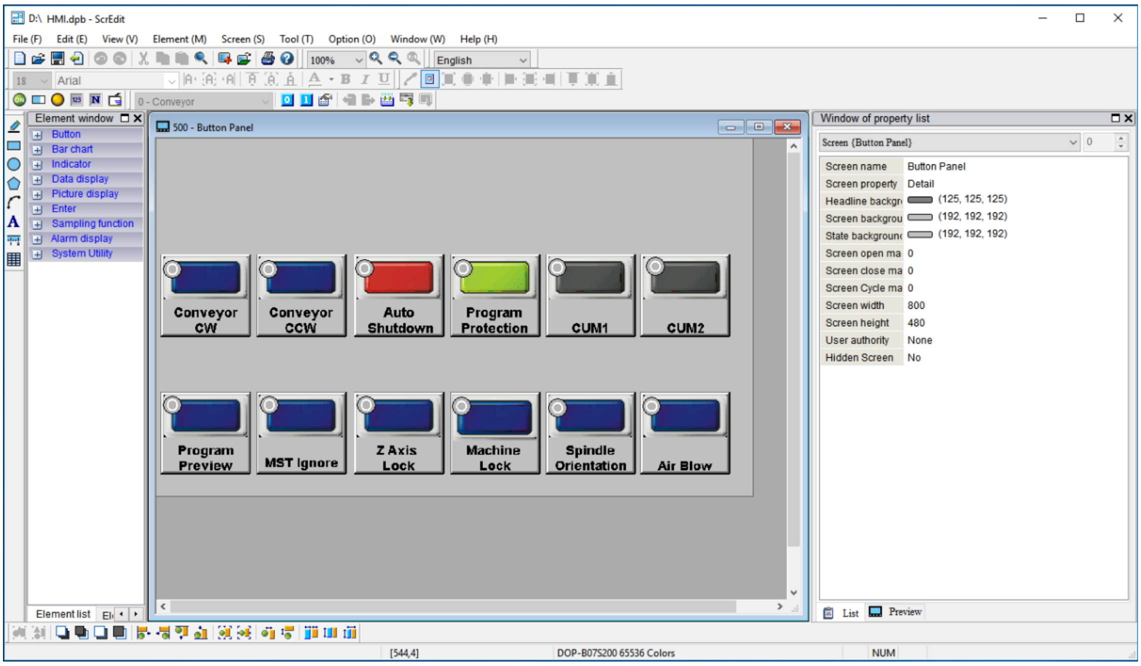


Figure 13.1.2

■ After compiling the screens and creating the screen data files, you can import the files to the controller using the USB disk or the Internet, as shown in Figure 13.1.3.

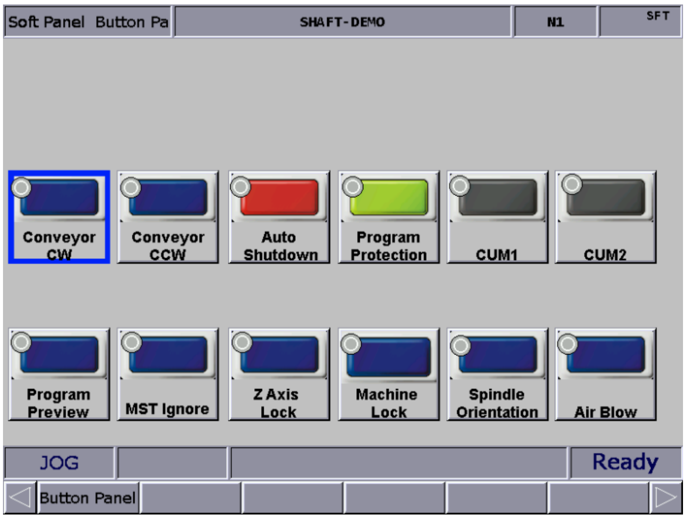


Figure 13.1.3

(This page is intentionally left blank.)

MLC Special M Relay and Special D Register 14

This chapter provides detailed descriptions for the special controlling devices of the NC system, through which you can quickly check the MLC status in the NC system. For more advanced control functions, refer to NC Series MLC Application Manual.

14.1 Introduction to MLC special M relay and special D register

The MLC (Motion Logic Control) and the NC are two independent systems. The MLC system performs button triggering controls, MLC axis movements, and other logic controls, while the NC system manages system and servo axis related functions. The MLC special M relays and D registers serve as the I/O interface between these two systems for data exchange and signal transmission.

Definition of output and input:

The output mentioned in this chapter refers to the signals sent to the NC system from the MLC special M relays and D registers. The input refers to the signals sent to the MLC special M and D from the NC system. The M letter prefixed codes are in bit format referring to signal 0 (OFF) or 1 (ON). The D prefixed codes are in word format referring to numerical values such as 1000. The MLC special M and D codes are all expressed in the form of M- and D- suffixed with four digits.

In the following paragraphs, the special M relays are abbreviated as special M and special D registers are abbreviated as special D.

Data exchanges between the two systems are categorized into four groups.

1: MLC bit output from MLC to NC (special M, bit output) (Refer to Chapter 14.2)

2: MLC bit input from NC to MLC (special M, bit input) (Refer to Chapter 14.3)

3: MLC word output from MLC to NC (special D, word output) (Refer to Chapter 14.4)

4: MLC word input from NC to MLC (special D, word input) (Refer to Chapter 14.5)

14.2 Special M relay bit output (from MLC to NC)

14.2.1 MLC and NC systems related special M output

When the special M status in the MLC system is changed, you can use the corresponding variable to access the status in the NC system. For example, #1801 is paired with M1024. If M1024 is on, the value of #1801 in the NC program is 1, and this value is 0 if M1024 is off.

Function name	Special M	Variable ID	Function name	Special M	Variable ID
HMI output point 1	M1024	#1801	HMI output point 17	M1040	#1817
HMI output point 2	M1025	#1802	HMI output point 18	M1041	#1818
HMI output point 3	M1026	#1803	HMI output point 19	M1042	#1819
HMI output point 4	M1027	#1804	HMI output point 20	M1043	#1820
HMI output point 5	M1028	#1805	HMI output point 21	M1044	#1821
HMI output point 6	M1029	#1806	HMI output point 22	M1045	#1822
HMI output point 7	M1030	#1807	HMI output point 23	M1046	#1823
HMI output point 8	M1031	#1808	HMI output point 24	M1047	#1824
HMI output point 9	M1032	#1809	HMI output point 25	M1048	#1825
HMI output point 10	M1033	#1810	HMI output point 26	M1049	#1826
HMI output point 11	M1034	#1811	HMI output point 27	M1050	#1827
HMI output point 12	M1035	#1812	HMI output point 28	M1051	#1828
HMI output point 13	M1036	#1813	HMI output point 29	M1052	#1829
HMI output point 14	M1037	#1814	HMI output point 30	M1053	#1830
HMI output point 15	M1038	#1815	HMI output point 31	M1054	#1831
HMI output point 16	M1039	#1816	HMI output point 32	M1055	#1832

14.2.2 NC system related special M output

The special M relays in Sections 14.2.2 - 14.2.4 are for signal transmission from the MLC to NC system. When you change the special M's status, the NC system operates accordingly.

You can use the keys or knob to send the special M signal from the MLC to the NC system and to switch between system modes or enable / disable the functions. Refer to the following table for the special M used for the NC system modes and functions.

<table>
<tr><th>Function name</th><th>Special M</th><th>Description</th></tr>
<tr><td>System mode selection:
0: auto execution (AUTO)
1: edit (EDIT)
2: manual input (MDI)
3: MPG feeding (MPG)
4: jog feeding (JOG)
5: rapid feeding (RAPID)
6: homing (HOME)</td><td>M1056
M1057
M1058
M1059</td><td>You can use M1056 - M1059 to switch between system modes.
<table>
<tr><th colspan="4">Binary</th><th rowspan="2">Decimal</th><th rowspan="2">System mode</th></tr>
<tr><th>M1059 (Bit 3)</th><th>M1058 (Bit 2)</th><th>M1057 (Bit 1)</th><th>M1056 (Bit 0)</th></tr>
<tr><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>AUTO</td></tr>
<tr><td>0</td><td>0</td><td>0</td><td>1</td><td>1</td><td>EDIT</td></tr>
<tr><td>0</td><td>0</td><td>1</td><td>0</td><td>2</td><td>MDI</td></tr>
<tr><td>0</td><td>0</td><td>1</td><td>1</td><td>3</td><td>MPG</td></tr>
<tr><td>0</td><td>1</td><td>0</td><td>0</td><td>4</td><td>JOG</td></tr>
<tr><td>0</td><td>1</td><td>0</td><td>1</td><td>5</td><td>RAPID</td></tr>
<tr><td>0</td><td>1</td><td>1</td><td>0</td><td>6</td><td>HOME</td></tr>
</table></td></tr>
<tr><td>Single block execution</td><td>M1060</td><td>In AUTO mode, program stops after one block is executed.</td></tr>
<tr><td>Cycle Start</td><td>M1061</td><td>Auto execution signal.</td></tr>
<tr><td>NC stop</td><td>M1062</td><td>The NC controller pauses after M1062 is triggered.</td></tr>
<tr><td>System stop</td><td>M1063</td><td>The system stops operating.</td></tr>
<tr><td>Dummy execution</td><td>M1065</td><td>After M1065 is triggered, the moving speed F of G01 in AUTO mode refers to the feedrate specified in the D1062 register.</td></tr>
<tr><td>Optional stop (M01 Pause)</td><td>M1066</td><td>Optional stop key. When the program executes M01, the controller immediately stops.</td></tr>
<tr><td>Single block skip ('/')</td><td>M1067</td><td>The program skips the block with symbol '/' when this function is enabled.</td></tr>
<tr><td>Lock all axes movements</td><td>M1068</td><td>Locks all axes movements of the machine (NC+MLC).</td></tr>
<tr><td>Lock Z axis movement</td><td>M1069</td><td>Locks the Z axis movement of the machine.</td></tr>
<tr><td>Ignore axis limit</td><td>M1070</td><td>The limit signal of each axis is ignored when this function is enabled.</td></tr>
<tr><td>Lock M, S, and T codes</td><td>M1071</td><td>Locks M, S, and T codes. The program skips M, S, and T codes in the execution.</td></tr>
<tr><td>DMCNET connection successful</td><td>M1072</td><td>The MLC sends this signal after the system confirms that the DMCNET connection is successful. Note that this signal signifies the connection is successful instead of the Servo On status.</td></tr>
<tr><td>Macro call initial preparation</td><td>M1074</td><td>The initial input of macro call (only works in AUTO mode and with correct macro ID).</td></tr>
<tr><td>Macro call activation</td><td>M1075</td><td>Activates macro call.</td></tr>
<tr><td>System reset</td><td>M1076</td><td>When M1076 is triggered, the NC system resets
(MLC > NC).</td></tr>
<tr><td>M99 stops</td><td>M1077</td><td>Set Pr308 [Bit 9] = 1 to disable the function of M99.
Set M1077 to on, and the NC system stops machining when it reads M99.</td></tr>
<tr><td>M96 program interruption</td><td>M1078</td><td>When the NC system reads M96, if M1078 is on, the system interrupts the main program and executes the subprogram.</td></tr>
<tr><td>MLC emergency stop</td><td>M1079</td><td>Pr305 [Bit 5] = 1: after M1079 is triggered, the system immediately stops.</td></tr>
<tr><td>MPG simulation</td><td>M1080</td><td>During program execution, you can use the MPG to control the speed of movement trajectories.</td></tr>
</table>

Function name	Special M	Description
Inhibit zone protection	M1085	Pr51 [Bit 5] = 1: enables the inhibit zone protection. When this special M is off, the inhibit zone protection is effective.
Notification of MST code execution complete	M1152	When M1152 is triggered, this signal informs the NC system that the M, S, or T codes have completed the execution.
Tool magazine 1 moves forward	M1168	Tool magazine 1 moves forward. When M1168 is triggered, the standby tool pot (D1373) value adds 1.
Tool magazine 1 moves backward	M1169	Tool magazine 1 moves backward. When M1169 is triggered, the standby tool pot (D1373) value minus 1.
Tool 1 exchange	M1170	Exchanges tool data in tool magazine 1. Exchange data between the tool number in use (D1374) and the standby tool number (D1371).
Tool magazine 1 reset	M1171	Triggers the tool magazine 1 reset function.
Tool magazine 2 moves forward	M1172	Tool magazine 2 moves forward. When M1172 is triggered, the standby tool pot (D1377) value adds 1.
Tool magazine 2 moves backward	M1173	Tool magazine 2 moves backward. When M1173 is triggered, the standby tool pot (D1377) value minus 1.
Tool 2 exchange	M1174	Exchanges tool data in tool magazine 2. Exchange data between the current spindle number (D1378) and the command tool number (D1375).
Tool magazine 2 reset	M1175	Triggers the tool magazine 2 reset function.
Panel MPG pulse +	M1118	Use the keys on the machine operation panel B as the MPG function and send forward pulse signals. Refer to the description for D1040 for the enabling method.
Panel MPG pulse -	M1119	Use the keys on the machine operation panel B as the MPG function and send reverse pulse signals. Refer to the description for D1040 for the enabling method.
Switch the spindle speed source	M1307	Off: the rotation speed command refers to the S code in the program. On: the rotation speed command refers to the value of D1148.
Breakpoint search	M1567	Triggers the breakpoint search function.
Lock User 1 permission	M2934	Locks User 1 permission. Set Pr10015 [Bit 1] to 1 (Account permission activation method) to use the M2934 function.
Restrict program editing	M2935	Restricts program editing of the controller.

14.2.3 NC axes related special M output

You can use the keys or knob to send the special M signal from the NC to the MLC system and to switch between system modes or enable / disable the functions. Refer to the following special M list for NC axis operations.

Function name	Special M	Function name	Special M
Trigger for synchronous control	M1088	Y axis homing control	M1237
X slave axis follows the master axis	M1089	Z axis homing control	M1238
Y slave axis follows the master axis	M1090	A axis homing control	M1239
Z slave axis follows the master axis	M1091	B axis homing control	M1240
A slave axis follows the master axis	M1092	C axis homing control	M1241
B slave axis follows the master axis	M1093	Cancel X axis 1st software limit	M1248
C slave axis follows the master axis	M1094	Cancel Y axis 1st software limit	M1249
Trigger for transition command	M1098	Cancel Z axis 1st software limit	M1250
X axis receives command from the master axis	M1099	Cancel A axis 1st software limit	M1251
Y axis receives command from the master axis	M1100	Cancel B axis 1st software limit	M1252
Z axis receives command from the master axis	M1101	Cancel C axis 1st software limit	M1253
A axis receives command from the master axis	M1102	Lock X axis	M1257
B axis receives command from the master axis	M1103	Lock Y axis	M1258
C axis receives command from the master axis	M1104	Lock Z axis	M1259
X axis forward jog control	M1216	Lock A axis	M1260
Y axis forward jog control	M1217	Lock B axis	M1261
Z axis forward jog control	M1218	Lock C axis	M1262
A axis forward jog control	M1219	X axis Servo Off	M1266
B axis forward jog control	M1220	Y axis Servo Off	M1267
C axis forward jog control	M1221	Z axis Servo Off	M1268
X axis reverse jog control	M1226	A axis Servo Off	M1269
Y axis reverse jog control	M1227	B axis Servo Off	M1270
Z axis reverse jog control	M1228	C axis Servo Off	M1271
A axis reverse jog control	M1229	U axis Servo Off	M1272
B axis reverse jog control	M1230	V axis Servo Off	M1273
C axis reverse jog control	M1231	W axis Servo Off	M1274
X axis homing control	M1236	-	-

14.2.4 Spindle related special M output

Refer to the following special M list for the spindle operation control.

Function name	Special M	Function name	Special M
Spindle forward rotation	M1120	Spindle positioning control	M1124
Spindle reverse rotation	M1121	Spindle retraction after tapping	M1125
Spindle gear ratio selection	M1122	Lathe spindle / C axis mode switch	M1126
	M1123	Spindle gain switch	M1127

Note:

The spindle gear ratio is selected by the combination of M1122 (Bit 0) and M1123 (Bit 1) with the range of 0 - 3 representing four sets of gear ratio (Pr422 - Pr429).

Example: to use the settings of “Gear ratio numerator 4 (Pr428)” and “Gear ratio denominator 4 (Pr429)”, select 3 (11 in binary format), and the corresponding two bits in the MLC are: M1122 = on and M1123 = on.

14

14.3 Special M relay bit input (from NC to MLC)

14.3.1 MLC and NC systems related special M input

You can write values to variables #1864 - #1895 in the NC program and then access and monitor the signal status of the HMI input points in the MLC system. For example, #1864 is paired with M2080. If you set #1864 to 1 in the NC program, M2028 is on in the MLC program. On the other hand, if you set #1864 to 0 in the NC program, M2028 is off in the MLC program.

Function name	Special M	Variable	Function name	Special M	Variable
HMI input point 1	M2080	#1864	HMI input point 17	M2096	#1880
HMI input point 2	M2081	#1865	HMI input point 18	M2097	#1881
HMI input point 3	M2082	#1866	HMI input point 19	M2098	#1882
HMI input point 4	M2083	#1867	HMI input point 20	M2099	#1883
HMI input point 5	M2084	#1868	HMI input point 21	M2100	#1884
HMI input point 6	M2085	#1869	HMI input point 22	M2101	#1885
HMI input point 7	M2086	#1870	HMI input point 23	M2102	#1886
HMI input point 8	M2087	#1871	HMI input point 24	M2103	#1887
HMI input point 9	M2088	#1872	HMI input point 25	M2104	#1888
HMI input point 10	M2089	#1873	HMI input point 26	M2105	#1889
HMI input point 11	M2090	#1874	HMI input point 27	M2106	#1890
HMI input point 12	M2091	#1875	HMI input point 28	M2107	#1891
HMI input point 13	M2092	#1876	HMI input point 29	M2108	#1892
HMI input point 14	M2093	#1877	HMI input point 30	M2109	#1893
HMI input point 15	M2094	#1878	HMI input point 31	M2110	#1894
HMI input point 16	M2095	#1879	HMI input point 32	M2111	#1895

14.3.2 NC system related special M input

You can get the NC system's current status with the signals sent from the NC system to the MLC special M. In addition, the MLC uses these signals as the feedback for status synchronization. The following table lists the NC system status and the corresponding MLC special M relays.

Function name	Special M	Description
Machine started and system ready	M2112	The NC system is ready.
System alarm message	M2113	A macro alarm occurs in the NC system. (MR__)
System emergency stop	M2114	The system stops immediately after you press the EMERGENCY STOP key.
Servo enabled	M2115	Servo ON.
HSI1	M2142	High speed input point 1 (G31 skip signal input).
HSI2	M2143	High speed input point 2 (G31 skip signal input).
M96 (program interruption) in operation	M2216	M96 (program interruption) is in operation.
Macro call initial preparation complete	M2224	Initialization setting for the macro call function is complete (only works in AUTO mode and with correct macro ID).
Macro call execution	M2225	Macro call execution.
Macro call error	M2226	Macro call error.
Synchronous function in execution	M2227	The NC system sends this signal when the synchronous function is in execution.
Transfer function in execution	M2228	The NC system sends this signal when the transfer function is in execution.
Special M for MPG forward	M2232	This special M relay is on when the MPG is in forward operation. This M relay is off when the MPG is in reverse operation or stationary.
M99 stop function	M2238	When the NC system reads M99, this special M relay is on.
Channel alarm message	M2240	An error occurs in the NC channel.
Auto execution (AUTO)	M2241	The NC sends this signal when it is in AUTO mode.
Edit (EDIT)	M2242	The NC sends this signal when it is in EDIT mode.
Manual input (MDI)	M2243	The NC system sends this signal when it is in MDI mode.
MPG feeding (MPG)	M2244	The NC system sends this signal when it is in MPG mode.
Jog feeding (JOG)	M2245	The NC system sends this signal when it is in JOG mode.
Rapid feeding (RAPID)	M2246	The NC system sends this signal when it is in RAPID mode.
Homing (HOME)	M2247	The NC system sends this signal when it is in HOME mode.
Single block execution	M2249	The NC system sends this signal when the program pauses after executing one single block.
Cycle Start	M2250	The NC system sends this signal when the program starts running.
Pause	M2251	The NC system sends this signal when the system pauses.
M00 program stop	M2252	The NC system sends this signal when executing M00.
M01 program stop (optional)	M2253	The NC system sends this signal when executing M01.
M02 end of program	M2254	The NC system sends this signal when executing M02.

Function name	Special M	Description
End of M30 and return	M2255	The NC system sends this signal when executing M30.
Program ends	M2271	The NC system sends this signal when the machining program ends.

14.3.3 M, S, T codes related special M input

When the M, S, or T codes are executed in the program, the NC system sends the corresponding special M signals to the MLC. For example, when M03 is executed in the NC program, the system sets M2208 to on in the MLC program accordingly. The following table lists the special M relays corresponding to the M, S, and T codes.

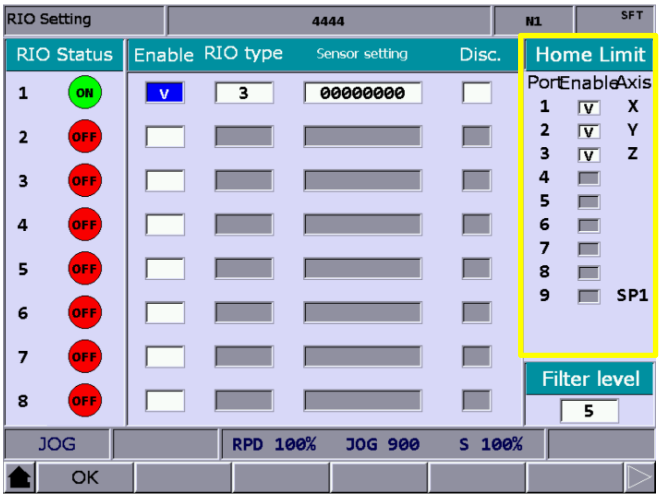
Function name	Special M	Description
M code execution	M2208	When the M code is executed in the program, the NC sends the corresponding special M signal to the MLC, so M2208 is set to on and instructs the specified device to operate accordingly. Until M1152 (Notification of MST code execution complete) is triggered, M2208 is set to off. When waiting for the MLC to return M1152, the NC pauses the program execution. After the MLC returns M1152, the NC resumes executing the program. The NC does not trigger M2208 when the program executes M00, M01, M02, M30, M98, M99, or the M codes that are used for macro call.
S code execution	M2209	When the S code is executed in the program, the NC sends the corresponding special M signal to the MLC, so M2209 is set to on and instructs the specified device to operate accordingly. Until M1152 (Notification of MST code execution complete) is triggered, M2209 is set to off. When waiting for the MLC to return M1152, the NC pauses the program execution. After the MLC returns M1152, the NC resumes executing the program. The NC does not trigger M2209 when an S code is used for macro call.
T code execution	M2210	When the T code is executed in the program, the NC sends the corresponding special M signal to the MLC, so M2210 is set to on and instructs the specified device to operate accordingly. Until M1152 (Notification of MST code execution complete) is triggered, M2210 is set to off. When waiting for the MLC to return M1152, the NC pauses the program execution. After the MLC returns M1152, the NC resumes executing the program. The NC does not trigger M2210 when a T code is used for macro call. M2210 is related to the station ID in the tool magazine. The NC system triggers M2210 only when the T code value is set within the specified range of tool number for the tool magazine parameter.

14.3.4 NC axes related special M input

There are three available methods for hardware signal input.

1. The AXIS 1~4 connector at the rear side of the controller*.
2. Remote I/O. Select the axes (as shown in the yellow mark in the figure), and the signals are input through RIO. The signals of the unselected axes are input through AXIS 1~4 connector.
3. Pr2006 [Bit 14]: origin / limit planning.

Note: the AXIS 1~4 connector is only available on the NC3XX series milling controllers.



The following table lists the special M relays corresponding to the hardware limits, homing signals, and axis movements of the NC axes.

Function name	Special M	Function name	Special M
Port 1 positive hardware limit	M2144	X axis homing complete	M2272
Port 1 negative hardware limit	M2145	Y axis homing complete	M2273
Port 1 home signal	M2146	Z axis homing complete	M2274
Port 2 positive hardware limit	M2148	A axis homing complete	M2275
Port 2 negative hardware limit	M2149	B axis homing complete	M2276
Port 2 home signal	M2150	C axis homing complete	M2277
Port 3 positive hardware limit	M2152	X axis positioned at the 2nd reference point	M2286
Port 3 negative hardware limit	M2153	Y axis positioned at the 2nd reference point	M2287
Port 3 home signal	M2154	Z axis positioned at the 2nd reference point	M2288
Port 4 positive hardware limit	M2156	A axis positioned at the 2nd reference point	M2289
Port 4 negative hardware limit	M2157	B axis positioned at the 2nd reference point	M2290
Port 4 home signal	M2158	C axis positioned at the 2nd reference point	M2291
Port 5 positive hardware limit	M2160	X axis is moving	M2320
Port 5 negative hardware limit	M2161	Y axis is moving	M2321
Port 5 home signal	M2162	Z axis is moving	M2322
Port 6 positive hardware limit	M2164	A axis is moving	M2323
Port 6 negative hardware limit	M2165	B axis is moving	M2324
Port 6 home signal	M2166	C axis is moving	M2325
X axis is moving in forward direction	M2336	X axis is moving in reverse direction	M2345
Y axis is moving in forward direction	M2337	Y axis is moving in reverse direction	M2346
Z axis is moving in forward direction	M2338	Z axis is moving in reverse direction	M2347

Function name	Special M	Function name	Special M
A axis is moving in forward direction	M2339	A axis is moving in reverse direction	M2348
B axis is moving in forward direction	M2340	B axis is moving in reverse direction	M2349
C axis is moving in forward direction	M2341	C axis is moving in reverse direction	M2350
U axis is moving in forward direction	M2342	U axis is moving in reverse direction	M2351
V axis is moving in forward direction	M2343	V axis is moving in reverse direction	M2352
W axis is moving in forward direction	M2344	W axis is moving in reverse direction	M2353
X axis positioned at the 3rd reference point	M2295	X axis positioned at the 4th reference point	M2368
Y axis positioned at the 3rd reference point	M2296	Y axis positioned at the 4th reference point	M2369
Z axis positioned at the 3rd reference point	M2297	Z axis positioned at the 4th reference point	M2370
A axis positioned at the 3rd reference point	M2298	A axis positioned at the 4th reference point	M2371
B axis positioned at the 3rd reference point	M2299	B axis positioned at the 4th reference point	M2372
C axis positioned at the 3rd reference point	M2300	C axis positioned at the 4th reference point	M2373
U axis positioned at the 3rd reference point	M2301	U axis positioned at the 4th reference point	M2374
V axis positioned at the 3rd reference point	M2302	V axis positioned at the 4th reference point	M2375
W axis positioned at the 3rd reference point	M2303	W axis positioned at the 4th reference point	M2376

14.3.5 Spindle, tool magazine, and MLC axes related special M input

During rigid tapping or before tool exchanges, you can use the following special M relays to check if the spindle is positioned and whether it reaches the target speed.

Function name	Special M	Function name	Special M
Spindle reaches the target speed	M2256	Spindle is in the rigid tapping mode	M2259
Spindle reaches zero speed	M2257	Rigid tapping interruption	M2260
Spindle positioning complete	M2258	Spindle homing complete	M2281

14.3.6 Lathe Spindle / C axis mode related special M input

Function name	Special M	Description
Lathe spindle / C axis mode	M2239	If the NC is in the Spindle mode and executes the M code for Spindle-to-C axis mode switch and M1126 is on, the NC switches from the Spindle to the C axis mode. After the switch is complete, M2239 is on and the NC is in C axis mode. If the NC is in the C axis mode and executes the M code for C axis-to-Spindle mode switch and M1126 is off, the NC switches from the C axis to the Spindle mode. After the switch is complete, M2239 is off and the NC is in Spindle mode. The system is in Spindle mode and M2239 is off by default.

14.4 Special D register output (from MLC to NC)

14.4.1 MLC and NC systems special D output

You can write values to the registers for HMI output points D1024 - D1039 in the MLC system and then access and monitor the values with variables #1833 - #1848 in the NC system. For example, #1833 is paired with D1024. If the value of D1024 in the MLC program is 100, the value of #1833 is 100.

Refer to the following table for the registers for the HMI output points in the MLC system and their corresponding variables in the NC system (MLC > NC):

Function name	Special D	Variable	Function name	Special D	Variable
HMI output point 1	D1024	#1833	HMI output point 9	D1032	#1841
HMI output point 2	D1025	#1834	HMI output point 10	D1033	#1842
HMI output point 3	D1026	#1835	HMI output point 11	D1034	#1843
HMI output point 4	D1027	#1836	HMI output point 12	D1035	#1844
HMI output point 5	D1028	#1837	HMI output point 13	D1036	#1845
HMI output point 6	D1029	#1838	HMI output point 14	D1037	#1846
HMI output point 7	D1030	#1839	HMI output point 15	D1038	#1847
HMI output point 8	D1031	#1840	HMI output point 16	D1039	#1848

14.4.2 NC system related special D output

The special D in this section is for data transmission from the MLC to the NC system. You can use the keys or control knob on the machine to have the MLC program output the special D value to the NC system, and to set the MPG function and change the feedrate. Refer to the following table for the special D functions.

Function name	Special D	Description
Number of the processing complete pieces (32-bit)	D1018	Set the value using D1022 in the Process screen or with the MLC input. If you set Pr10015 [Bit 6] to 0, the display source is D1022. If you set Pr10015 [Bit 6] to 1, the display source is D1018 (D1019).
Number of the processing target (32-bit)	D1020	Set the value using D1023 in the Process screen or with the MLC input. If you set Pr10015 [Bit 6] to 0, the display source is D1023. If you set Pr10015 [Bit 6] to 1, the display source is D1020 (D1021).
Number of the processing complete pieces	D1022	Set the value using D1022 in the Process screen or with the MLC input.
Number of the processing target	D1023	Set the value using D1023 in the Process screen or with the MLC input.
MPG operation mode number	D1040	Sets the MPG operation mode. Set D1040 to 0 to use the external MPG. Set D1040 to 10 to use the keys on the machine operation panel B as the MPG function, with the pulse control trigger flags of M1118 and M1119.
MPG operation channel selection	D1041	Designates the MPG operation channel. The default value is 0.
Sets the MPG pulse magnification	D1042	Sets the MPG pulse magnifications. It usually works with the physical MPG. When you rotate one MPG scale, the actual movement is the minimum unit multiplied by the pulse magnification. For example, if you set D1042 to 1 and set Pr301 to 3 (set the minimum unit with three decimal places), the actual moving amount is 1 x 0.001 = 0.001 mm/pulse.
Select the axis with MPG	D1043	Selects the axis to move with the MPG. Set 0 for X axis, 1 for Y axis, and 2 for Z axis.
Macro call file name	D1111	Specifies the macro call file name O9xxx. For example, if D1111 writes K9100, the system calls the macro named O9100.
Lathe tool number selection	D1115	When the lathe parameter Pr308 [Bit 13] is 1, D1115 determines the lathe tool number display. Range: 0 - 65535.
Spindle speed command	D1148	When M1308 is on, the spindle speed command refers to the setting of D1148.

14

14.4.3 NC axes related special D output

The MLC uses the special D registers to send signals to the NC system, so it can change the feedrate in each NC system mode. Refer to the following table for the special D functions.

Function name	Special D	Description
Cutting feedrate adjustment	D1056	Sets the percentage of the cutting feedrate (F) in the NC program. Example: If you set F to 1000 and the current value of D1056 is 50, it means the actual command speed is 500 mm/min (= 1000 x 50%).
Rapid movement speed adjustment	D1058	Sets the percentage of G00's speed (rapid movement). For example, if the rapid movement speed is 6000, and D1058 is set to 50, it means the actual speed of G00 is 3000 mm/min (= 6000 x 50%).
Spindle speed adjustment	D1060	Sets the percentage for the S value specified in the NC program. For example, if S1000 is given in the program and D1060 is set to 30, it means the actual spindle speed is 300 rpm (= 1000 x 30%).
Sets the jog feeding and dry run speed	D1062	Sets movement speed F for dry run in JOG or AUTO mode. If you set D1062 to 50, it indicates F50 (mm/min) with the range of 0 - 65535 mm/min.

14.5 Special D register input (from NC to MLC)

14.5.1 MLC and NC systems related special D input

You can write the values to variables #1896 - #1911 in the NC program and then access and monitor the values in the registers for HMI input points in the MLC system. For example, #1896 is paired with D1336. If the value of D1336 in the MLC program is 101, the value of #1896 is 101. In other words, the value of D1336 in the MLC program changes with #1896 in the NC system.

Refer to the following table for the registers for HMI input points in the MLC system and their corresponding variables in the NC system (NC > MLC):

Function name	Special D	Variable	Function name	Special D	Variable
HMI input point 1	D1336	#1896	HMI input point 9	D1344	#1904
HMI input point 2	D1337	#1897	HMI input point 10	D1345	#1905
HMI input point 3	D1338	#1898	HMI input point 11	D1346	#1906
HMI input point 4	D1339	#1899	HMI input point 12	D1347	#1907
HMI input point 5	D1340	#1900	HMI input point 13	D1348	#1908
HMI input point 6	D1341	#1901	HMI input point 14	D1349	#1909
HMI input point 7	D1342	#1902	HMI input point 15	D1350	#1910
HMI input point 8	D1343	#1903	HMI input point 16	D1351	#1911

14

14.5.2 M, S, T codes related special D input

When the M, S, and T codes are executed in the program, the NC system sends the corresponding special M signals to the MLC and inputs the corresponding values to the special D registers. For example, when the NC program is executing M03, D1368 displays 3 in the MLC program. Refer to the following table for descriptions.

Function name	Special D	Description
M code data	D1368	When an M code is executed in a program, the index value of the M code is stored in register D1368. For example, when M03 command is executed, the value of D1368 is 3. The M codes mentioned here do not include the system-defined M codes such as M00, M01, M02, M30, M96, M97, M98, M99, and the M codes used for macro call. If one of these M codes are executed, the system does not write the value to D1368.
S code data	D1369	When an S code is executed in a program, the index value of the S code is stored in register D1369.
T code data (command)	D1370	When a T code is executed in a program, the index value of the T code is stored in register D1370. When the T code has been used for macro call, the system does not write the value to D1370. You must set the T code value within the specified range of tool number for the tool magazine parameter (Pr338 or Pr341) for the T data to correctly display.
Standby tool number (tool magazine 1)	D1371	The Register magazine in tool magazine 1 displays the tool number corresponding to the standby tool pot (D1373).
Tool pot deviation (tool magazine 1)	D1372	The deviation between the positions specified in D1370 (T code data) and D1371 (Standby tool number) in tool magazine 1. When the tool magazine rotates forward or backward during tool exchange (M1168 / M1169), the current tool magazine needs to rotate according to the value in D1372 for compensating the offset.
Standby tool pot (tool magazine 1)	D1373	The standby tool pot number in tool magazine 1.
Tool number in use (tool magazine 1)	D1374	The tool number in use in tool magazine 1.
Standby tool number (tool magazine 2)	D1375	The Register magazine in tool magazine 2 displays the tool number corresponding to the standby tool pot (D1377).
Tool pot deviation (tool magazine 2)	D1376	The deviation between the positions specified in D1370 (T code data) and D1375 (Standby tool number) in tool magazine 2. When the tool magazine rotates forward or backward during tool exchange (M1172 / M1173), the current tool magazine needs to rotate according to the value in D1376 for compensating the offset.
Standby tool pot (tool magazine 2)	D1377	The standby tool pot number in tool magazine 2.
Tool number in use (tool magazine 2)	D1378	The tool number in use in tool magazine 2.
Feedrate	D1379	Accesses the feedrate during cutting.
Spindle actual speed	D1380	You can use D1380 (32-bit) to access the spindle actual speed. You can use Pr399 [Bit 12] to change the value displaying source. Pr399 [Bit 12] = 0: the source is the S code command in the program. Pr399 [Bit 12] = 1: the source is the spindle current speed.
Workpiece coordinate group	D1450	Displays the current workpiece coordinates.

14.5.3 NC axes related special D input

The NC system sends the special D signals to the MLC system. And you can use the special D to access the data such as machine coordinates, absolute coordinates, and current percentage through the MLC. The descriptions are as follows.

Function name	Special D	Description
X axis machine coordinate	D1384	X axis current machine coordinate.
Y axis machine coordinate	D1386	Y axis current machine coordinate.
Z axis machine coordinate	D1388	Z axis current machine coordinate.
A axis machine coordinate	D1390	A axis current machine coordinate.
B axis machine coordinate	D1392	B axis current machine coordinate.
C axis machine coordinate	D1394	C axis current machine coordinate.
U axis machine coordinate	D1396	U axis current machine coordinate.
V axis machine coordinate	D1398	V axis current machine coordinate.
W axis machine coordinate	D1400	W axis current machine coordinate.
X axis absolute coordinate	D1402	X axis current absolute coordinate.
Y axis absolute coordinate	D1404	Y axis current absolute coordinate.
Z axis absolute coordinate	D1406	Z axis current absolute coordinate.
A axis absolute coordinate	D1408	A axis current absolute coordinate.
B axis absolute coordinate	D1410	B axis current absolute coordinate.
C axis absolute coordinate	D1412	C axis current absolute coordinate.
U axis absolute coordinate	D1414	U axis current absolute coordinate.
V axis absolute coordinate	D1416	V axis current absolute coordinate.
W axis absolute coordinate	D1418	W axis current absolute coordinate.
X axis DMCNET current monitoring	D1420	X axis current percentage at present.
Y axis DMCNET current monitoring	D1421	Y axis current percentage at present.
Z axis DMCNET current monitoring	D1422	Z axis current percentage at present.
A axis DMCNET current monitoring	D1423	A axis current percentage at present.
B axis DMCNET current monitoring	D1424	B axis current percentage at present.
C axis DMCNET current monitoring	D1425	C axis current percentage at present.
U axis DMCNET current monitoring	D1426	U axis current percentage at present.
V axis DMCNET current monitoring	D1427	V axis current percentage at present.
W axis DMCNET current monitoring	D1428	W axis current percentage at present.
SP1 DMCNET current monitoring	D1429	Spindle 1 current percentage at present.
SP2 DMCNET current monitoring	D1430	Spindle 2 current percentage at present.

14

14.6 NC / MLC axis mode switch

You can use the special M to dynamically switch between the NC axis and MLC axis modes, and control the position and speed in the MLC axis mode with special M and special D.

NC / MLC axis mode switch:

Axis	Special M for NC / MLC axis switch	Special M for NC / MLC axis status
X axis	M1200	M2354
Y axis	M1201	M2355
Z axis	M1202	M2356
A axis	M1203	M2357
B axis	M1204	M2358
C axis	M1205	M2359
U axis	M1206	M2360
V axis	M1207	M2361
W axis	M1208	M2362

MLC axis mode control / status:

Axis	Special M for activating MLC axis	Special M for absolute / incremental command	Special M for switching between MLC position / speed mode	Special D for positioning command	Special D for feed command	Special M for indicating position / speed reached	Special M for indicating the axis is moving
X axis	M1184	M1280	M1289	D1064	D1082	M2304	M2320
Y axis	M1185	M1281	M1290	D1066	D1084	M2305	M2321
Z axis	M1186	M1282	M1291	D1068	D1086	M2306	M2322
A axis	M1187	M1283	M1292	D1070	D1088	M2307	M2323
B axis	M1188	M1284	M1293	D1072	D1090	M2308	M2324
C axis	M1189	M1285	M1294	D1074	D1092	M2309	M2325
U axis	M1190	M1286	M1295	D1076	D1094	M2310	M2326
V axis	M1191	M1287	M1296	D1078	D1096	M2311	M2327
W axis	M1192	M1288	M1297	D1080	D1098	M2312	M2328
ALL	-	M1194	-	-	-	-	-

Troubleshooting 15

This chapter provides the information about the alarms and troubleshooting methods for the NC system. Search this chapter for the methods of handling the NC system related malfunctions.

15.1 Alarm description

The CNC alarms can be divided into three categories, which are system alarms, user-defined alarms, and user-defined macro alarms. This chapter mainly explains the system related alarms while the rest are user-defined.

Alarm category	Alarm code	Alarm Description
System alarms	_ _ _ _	The system alarms caused by system error or operation error.
User-defined alarms	A_	The user-defined alarms which you use with the MLC program. When the A_ device is triggered, the alarm corresponding to the A_ device also occurs.
User-defined macro alarms	MR_	The user-defined macro alarm works with variable #6000. You can have the alarm triggered and have the corresponding macro alarm displayed on the controller.

The system alarms are divided into MLC related alarms, HMI related alarms, and NC related alarms by function.

System alarm category	Alarm code range	Subcategory	Description
MLC related alarms	1200 - 12FF	MLC system errors	-
	1300 - 13FF	Ethernet errors	-
	1E00	Servo errors	This alarm displays the information about the servo station number in error and error code.
	1F00	Remote I/O errors	This alarm displays the information about the remote I/O station number in error.
HMI related alarms	3010 - 3FFF	HMI errors	-
NC related alarms	4200 - 4FFF	NC system errors	-
	B000 - EFFF	NC channel errors	According to the high-byte of the alarm code, you can identify which channel is in error: BXXX: CH0 CXXX: CH1 (reserved) DXXX: CH2 (reserved) EXXX: CH3 (reserved) These types of errors are also displayed differently according to functions. Some alarms carry data and others don't. Thus, you can use the alarm code to determine whether the alarm carries data or not.

15.1.1 MLC errors (1200 - 13FF)

Alarm code	Name	Cause and corrective action
1200	NC memory access error NC MEMORY ERROR	An error occurs when the MLC accesses the NC memory.
		Restart the controller or send it back for servicing.
1201	NC not ready NC NOT READY	The startup of the NC system is not complete.
		Restart the controller or send it back for servicing.
1202	Memory error INCORRECT NC BUF DATA	NC memory buffer error or not ready.
		Restart the controller or send it back for servicing.
1203	Output port does not exist NC PORT NOT EXIST	NC output port does not exist.
		Check the axis parameter setting.
1204	MLC code clear error MLC CODE ERASE FAIL	Failed to clear the MLC codes.
		Send the controller back for servicing.
1205	MLC flash memory error MLC CODE FLASH FAIL	Failed to write the MLC codes to the flash memory.
		Restart the controller or send it back for servicing.
1206	SRAM error SRAM FAIL	SRAM write-in error.
		Send the controller back for servicing.
1207	Host I/O channel error ON BOARD I/O ACCESS ERROR	An error occurs when the system accesses the host I/O.
		Restart the controller or send it back for servicing.
1208	Remote I/O channel error REMOTE I/O ACCESS ERROR	An error occurs when the system accesses the remote I/O.
		Restart the controller or send it back for servicing.
120A	NC parameter error NC PARAMETER EMPTY	NC parameters are not set or initialized.
		Re-initialize the parameters.
120B	Compensation PAR error COMP INIT ERROR	Compensation parameter write-in error.
		Reload the compensation parameters.
120C	Compen. PAR clear error COMP MEM ERASE FAIL	An error occurs when the system clears the compensation parameter in memory.
		Reload the compensation parameters.
120D	Compen. PAR write-in error COMP MEM WRITE FAIL	An error occurs when the system writes the compensation parameters to memory.
		Reload the compensation parameters.
120E	PAR initialization error DEFAULT INIT ERROR	Parameter initialization error.
		Re-initialize the parameters.
120F	Memory clear error DEFAULT MEM ERASE FAIL	Memory clear error.
		Restart the controller or send it back for servicing.
1210	Memory write-in error DEFAULT MEM WRITE FAIL	Memory write-in error or memory initialization error.
		Restart the controller or send it back for servicing.
1211	Servo axis does not exist SRV AXIS NOT EXIST	Parameter setting error.
		Check the parameter settings.
1212	Servo axis PAR type error SRV TYPE NOT EXIST	Parameter setting error.
		Check the parameter settings.
1213	DMCNET initialization error DMCNET INIT ERROR	DMCNET initialization error.
		Make sure the DMCNET cable is securely connected.
1214	Non-volatile memory error FRAM FAIL	Non-volatile memory error.
		Restart the controller or send it back for servicing.
1215	Internal comm. error INTERNAL COMM. ERROR	MLC and NC bottom layer communication error.
		Use the correct firmware version. Set the correct channel parameters.

Alarm code	Name	Cause and corrective action
1216	MLC PRG error MLC CODE ERROR	The following conditions occur when the MLC is in execution: The divisor of the division command is 0. The jump target position of the CJ/CALL command is invalid. The number of digits displayed in the BCD command is invalid. The table number of the VRT command is invalid. MLC code error.
		Check and correct the MLC program. Reload the MLC program.
1217	MLC PAR does not match MLC PAR MISMATCH	Current number of MLC parameters in the internal memory do not match the number of parameters planned in the firmware.
		Update firmware or send the controller back for servicing.
1300	Network com. error ETHERNET ERROR	Check the network connection.
		Restart the controller or send it back for servicing.

15.1.2 Servo errors (1E00)

Alarm code	Name	Cause and corrective action
1E00	Servo error SERVO ERROR	[1530]: alarm signal (DI) on the pulse card is triggered. Check the error cause of the connected drive.
		[1531]: this alarm occurs when the pulse card generates more than 24,000 pulses within 1 ms. Check the controller's encoder pulse number setting and make sure the motor speed is within the range.
		[1532]: pulse leakage. When the motor is stopped, the pulse card compares the command pulse number with the feedback pulse number. If the pulse number difference is greater than 1% of the encoder resolution, this alarm occurs. Check if the signal traces are shielded. Check if the equipment is properly grounded.
		[1560]: station number repetition error. Check the channel parameter station number setting. Check the pulse card station number switch knob setting. Check the drive station number P3-00 setting.
		Refer to the relevant servo drive user manuals for the definition and troubleshooting of other related errors.

15.1.3 Remote I/O error (1F00)

Alarm code	Name	Cause and corrective action
1F00	Remote I/O error REMOTE I/O ERROR	Remote I/O error.
		Check the remote I/O connection or replace the remote I/O board.

15.1.4 HMI related alarms (3010 - 3FFF)

Alarm code	Name	Cause and corrective action
3010	HMI communication interface establishing error INTERFACE RX CREATE ERROR	An error occurs when the system establishes the HMI communication interface.
		Restart the controller or send it back for servicing.
3011	HMI communication memory area creating error RX THREAD CREATE ERROR	An error occurs when the system creates the HMI communication memory area.
		Restart the controller or send it back for servicing.
3012	HMI interface command area error IF CMD THREAD CREATE ERROR	An error occurs when the system creates the HMI interface command area.
		Restart the controller or send it back for servicing.
3013	HMI interface memory area error SRAM VERIFY ERROR	HMI interface memory area error.
		Restart the controller or send it back for servicing.
3014	HMI interface communication port error INTERFACE RX PORT ERROR	HMI interface communication port error.
		Restart the controller or send it back for servicing.
3015	MLC interface memory area error MLC SRAM VERIFY ERROR	MLC interface memory area error.
		Restart the controller or send it back for servicing.
3016	HMI file transmission error INVALID REQUEST FILE	HMI file transmission error.
		Restart the controller or send it back for servicing.
3017	HMI data transmission error REQUEST FILE MISMATCH	HMI data transmission error.
		Restart the controller or send it back for servicing.
3018	FTP PAR setting error INVALID FTP PARAMETER	Incorrect FTP setting.
		Reconfirm FTP related settings of the controller.
3100	Invalid file name INVALID FILE NUMBER	Invalid file name.
		Modify the file name.
3101	Exceeded subroutine call LVL EXCEED MAX FILE LEVEL	The macro call exceeded 8 layers.
		Check the macro program.
3102	Non-G code character error ILLEGAL GCODE CHAR	Non-G code character error.
		Check the G code and modify the program.
3103	Memory error MEM CHECKSUM ERROR	System internal memory area is in error.
		Use the system recovery function or send the controller back for servicing.
3200	Internal PAR CRC error PARAMETER CRC ERROR	System internal parameter memory area is in error.
		Use the system recovery function or send the controller back for servicing.
3201	MLC PRG error MLC CRC ERROR	System MLC program memory area is in error.
		Reload the MLC program or send the controller back for servicing.
3202	CF card read failed CF CARD READ ERROR	No CF card is inserted or the inserted CF card is invalid.
		Make sure the CF card is correctly installed or replace the CF card.

Alarm code	Name	Cause and corrective action
3203	PAR backup failed PAR BACKUP FILE ERROR	The CF card is not correctly inserted or does not have sufficient storage space.
		Make sure the CF card is correctly installed or check the CF card storage capacity.
3204	MLC backup failed MLC BACKUP FILE ERROR	The CF card is not correctly inserted or does not have sufficient storage space.
		Make sure the CF card is correctly installed or check the CF card storage capacity.
3205	Machine is locked CNC MACHINE LOCK	The usage duration has expired.
		Contact the equipment manufacturer to unlock or extend the machine usage duration.
3206	PAR value exceeds the range PARAMETER VALUE OUT OF RANGE	There are system parameters with set values exceeding the allowable range.
		Check and modify the parameter values.
3207	COM port DLL load error LOAD COMM DLL ERR	Failed to load the external device driver with the software panel.
		Update the software panel to reload the external device driver.
3208	Machine to be locked soon MACHINE WILL BE LOCKED	System usage duration expiring reminder.
		Contact the equipment manufacturer to unlock or extend the machine usage duration.
3209	Sys updated, please reboot UPDATE FINISHED, PLEASE REBOOT SYS	The system update is complete and requires restarting.
		Restart the controller.
320A	System battery is low SYSTEM LOW BATTERY	The system battery is low.
		Replace the controller battery.
3210	COM port disconnection COMM PORT DISCONNECT	The connection between the software panel and the external device is disconnected.
		Check the connection settings between the controller and the external device.
3211	COM port DLL open error OPEN COMM DLL ERR	Failed to open the external device driver with the software panel.
		Update the software panel to reload the external device driver.
3212	Failed to create COM port COM PORT CREATE FAIL	An error occurs when the external device driver is loaded with the software panel.
		Update the software panel and make sure the setting of the external link file is correct.
3213	Load TCPIP DLL Err TCPIP LOAD DLL FAIL	Failed to load the TCPIP driver with the software panel.
		Update the software panel to reload the external device driver.
3214	TCPIP comm. disconnection TCPIP DISCON	Network connection is in error.
		Check the network connection or connection status.
3215	Network comm. init. failed ETHERNET INIT FAIL	Failed to initialize the network communication.
		Check the network connection or connection status.
3216	System reset error SYSTEM RESET FAIL	No response from the bottom layer after resetting.
		Update the firmware or send the controller back for servicing.
3217	Failed to import full backup FAIL TO IMPORT FULL BACKUP	Failed to remotely enable the backup import.
		Check the operating mode. Make sure the backup storage device is correctly installed.

Alarm code	Name	Cause and corrective action
3218	Failed to export full backup FAIL TO EXPORT FULL BACKUP	Failed to remotely enable the backup export.
		Check the operating mode. Make sure the backup storage device is correctly installed.
3219	Auto update is set, please reboot AUTO UPDATE IS SET, PLEASE REBOOT SYS	Failed to remotely enable the firmware update function.
		Cycle the power to start the automatic firmware update function. You must put the firmware file in the [pkt] folder in the root directory of the USB disk.
3220	Sys. update failed, please reboot UPDATE FAIL, PLEASE REBOOT SYS	Failed to update the remote system firmware.
		After checking the following conditions, reset the automatic firmware update function. Firmware update model error. File format does not match. Firmware update version is not supported. Controller emergency stop is not pressed.

15.1.5 NC system errors (4200 - 4FFF)

Alarm code	Name	Cause and corrective action
4200	Execute homing EXECUTE HOME RETURN	Homing has not been executed for each axis before automatic operation.
		Execute homing for each axis.
4201	Absolute origin setting RESET ABS	The origin coordinates for the absolute type motor are not established.
		Go to [DGN] to execute **ABS RST**.
4300	MLC is not ready MLC ACCESS MEM ERROR	An error occurs when the MLC accesses the NC memory.
		Restart the controller, update firmware, or send the controller back for servicing.
4301	MLC is not ready HOST NOT READY	The MLC is not ready.
		Restart the controller, update firmware, or send the controller back for servicing.
4302	I/O module program clearing failed FPGA ERASE ERROR	Failed to clear the I/O module program.
		Update the I/O module firmware or send the controller back for servicing.
4303	I/O module program write-in failed FPGA FLASH ERROR	Failed to write the I/O module program.
		Update the I/O module firmware or send the controller back for servicing.
4304	NC system program clearing failed PROG ERASE ERROR	Failed to clear the NC system program.
		Restart the controller, update firmware, or send the controller back for servicing.
4305	NC system program installation failed PROG FLASH ERROR	Failed to install the NC system program.
		Restart the controller, update firmware, or send the controller back for servicing.
4306	Macro clearing failed MACRO ERASE ERROR	Failed to clear the macro program.
		Restart the controller and execute the macro internal memory function.
4307	Macro installation failed MACRO FLASH ERROR	Failed to install the macro program.
		Restart the controller and execute the macro internal memory function.

Alarm code	Name	Cause and corrective action
4308	G code loading error ILLEGAL PROGRAM COUNT	An error occurs when the system loads the G code. Check the machining program.
4310	I/O module PRG not initialized FPGA CODE NOT EXIST	The I/O module program is not initialized. Update the I/O module firmware or send the controller back for servicing.
4311	I/O module memory corrupted FPGA CODE LENGTH ERROR	The I/O module memory is corrupted. Update the I/O module firmware or send the controller back for servicing.
4312	I/O module memory corrupted FPGA CODE CORRUPTED	The I/O module memory is corrupted. Update the I/O module firmware or send the controller back for servicing.
4313	I/O module status error FPGA STATUS REPLY ERROR	The I/O module status is in error. Check if the I/O board is firmly installed and the wiring is securely connected.
4314	I/O module PRG configuration error FPGA CONF DONE ERROR	Program configuration of the I/O module is in error. Check if the I/O board is firmly installed and the wiring is securely connected.
4315	I/O board hardware interface error FPGA IF ERROR	The I/O board hardware interface is in error. Update the I/O module firmware or send the controller back for servicing.
4316	I/O board hardware interface error FPGA BUS ERROR	An error occurs when the system reads the I/O board hardware interface. Update the I/O module firmware or send the controller back for servicing.
4317	NC system command error ILLEGAL COMMAND	The NC system command is in error. Check the machining program or the macro content.
4318	NC parameter error NC PARA NOT LOAD	The NC parameter does not exist or the MLC is not ready. Restart the controller, update firmware, or send the controller back for servicing.
4319	NC parameter error NC PARA NOT EXIST	The NC parameter does not exist or the MLC is not ready. Restart the controller, update firmware, or send the controller back for servicing.
431A	Tool magazine axis error ATC UNDEFINED CONFLICT	The tool magazine axis number is not defined or defined repeatedly. Check the parameter settings.
431B	NC parameter error INVALID NC PARA	The NC parameter is in error. Make sure no parameter values are set to 0, such as the encoder pulse number, the gear numbers of the output shaft and motor end.
431C	Spindle polarity error SPINDLE POLARITY ERROR	Spindle voltage output does not match the motor rotation direction. Check if the spindle encoder OA/OB wiring is correct. By setting Bit 6 of Pr51, you can adjust the OA/OB phase sequence of the spindle encoder feedback.

Alarm code	Name	Cause and corrective action
4FFC	Servo overflow protection SERVO OVERFLOW PROTECTION	This alarm occurs when the Servo On / Off action is executed in AUTO mode and the position overflow occurs on the specified axis, so the system does not execute this Servo On / Off action.
		Check the servo parameters for preventing position overflow.
4FFD	Position feedback protection POSITION FEEDBACK PROTECTION	When the motion command is issued, the position feedback did not change.
		Check the servo settings, such as the torque limit is too low, bandwidth is too low, or the maximum rotation speed is too low. Check if the servo feedback signal wiring functions normally. Check if the motor rotates normally.
4FFE	Overspeed protection UNEXPECTED ACC. PROTECTION	There is an overspeed motion command.
		Make sure the parameter settings are not set too high, such as the output gear ratio and cutting speed setting.

15.1.6 Machining related alarms (B000 - B0FF)

Alarm code	Name	Cause and corrective action
B000	Illegal G code line number ILLEGAL GCODE LINE NUMBER	The G code line number is illegal. Check the machining program.
B001	Illegal G code length ILLEGAL GCODE LENGTH	The G code length is illegal. Check the machining program.
B002	G code file not found GCODE FILE NOT FOUND	The G code file is not found. Check the file contents.
B003	Invalid file name loaded NO SPECIFIC O FILE	The name of the loaded file is invalid. Reload the program file.
B004	Workpiece coordinates computing error PPO WRK COORD OVERLAP	An error occurs when the system computes the workpiece coordinates. Reset the workpiece coordinates.
B005	Workpiece coordinates computing error PPO WRK COORD EMPTY	An error occurs when the system computes the workpiece coordinates. Reset the workpiece coordinates.
B006	Workpiece coordinates computing error PPO WRK COORD MISMATCH	An error occurs when the system computes the workpiece coordinates. Reset the workpiece coordinates.
B007	Servo port setting conflict SERVO PORT CONFLICT	There is a conflict in the servo port settings. Check the parameter settings.
B008	Memory overlap PPO G BUF OVERLAP	An error occurs when the system internal program is operating. Reload the machining program or update the firmware.
B009	G code buffer error PPO G BUF EMPTY	The G code buffer is in error. Reload the machining program or update the firmware.
B00A	Interpolator cmd index Err PPO CMD IDX MISMATCH	An error occurs to the computing interpolator command index in the system. Reload the machining program or update the firmware.
B00B	INTRPL cmd BUF access Err PPO CMD BUF EMPTY	An error occurs when the system accesses the buffer of the computing interpolator command. Reload the machining program or update the firmware.
B00C	Undefined feedrate UNASSIGNED FEEDRATE	The feedrate is undefined. Check the machining program and define the feedrate.
B00D	Arc radius error RADIUS MISMATCH	Arc radius calculation is in error. Check the arc machining program or increase the tolerance for arc radius error.
B00E	Tool No. selection Err INVALID CUTTER	The specified tool number is in error during tool change or T code execution. Check the changed tool number and if the T code setting is within the tool setting range.
B00F	Servo connection axes Err SERVO CONNECTION ERROR	The number of the connection axes does not match the parameter setting. Check the set channel port number and the connecting servo station number setting.
B010	Breakpoint No. not found SEARCH CONFLICT	Breakpoint search is assigned to M99. Reset the breakpoint search line number.

Alarm code	Name	Cause and corrective action
B014	Incorrect tool compen. radius TOOL COMPENSATION RADIUS ERROR	The system cannot correctly calculate the tool radius compensation value specified in the G code.
		Check the G codes and modify the program.
B015	Sync cmd error INVALID SYNCHRONOUS COMMAND	This alarm occurs when the synchronous function is enabled and you use the G code to specify the slave axis moving amount.
		Check the machining program. After the synchronous function is enabled, do not specify the slave axis moving amount in the G code.
B017	Tool assignment error INVALID TOOL ASSIGNMENT	The tool compensation number in the G code exceeds the number range.
		Adjust the tool compensation number setting in the G code.
B018	Cmd transfer not allowed INVALID TRANSITION CHANGE	An error occurs when the system checks for the command transfer in MDI mode.
		Command transfer can only be done before the machining program is executed.
B019	Servo command error INVALID SERVO COMMAND	The axis does not receive G code motion commands when the servo is off.
		Check if the command axis is in the Servo Off state.
B01A	Data amount error MESSAGE BOX FULL	The processing amount of the interface data exceeds the range, such as tool changing, magazine setting, variable writing (#_), and G10 data setting.
		Check if the G codes and MLC processing actions are operating normally.
B01B	Spindle not running SPINDLE IS NOT RUNNING	The spindle is not rotating during machining. Pr51 [Bit 0]: the function for checking the spindle before cutting is enabled.
		Make sure the spindle rotation command is issued.
B01C	Spindle cmd speed error ABNORMAL SPINDLE COMMAND	The spindle command exceeds the spindle maximum speed.
		Redefine the spindle speed. Adjust Pr409 for the spindle maximum speed setting.
B01D	Stroke limit error STROKE LIMIT ERROR	The stop function is triggered when the path enters the restricted area.
		AUTO mode: correct the G code execution path. MDI mode: move to the opposite direction of the restricted area to clear the error.
B01F	Spindle feedback error ABNORMAL SPINDLE FEEDBACK	The spindle is in analog voltage and closed-loop control. This alarm occurs when the spindle encoder feedback is in error. Note: the analog output voltage reduces to 0V after this alarm occurs.
		Check if the wiring for the encoder cable is correct.
B020	Emergency stop EMERGENCY STOP	An emergency stop occurs.
		Check if the EMG button is pressed. Check the emergency stop status.
B021	Chamfer / rounding Err CHAMFERING / ROUNDING ERROR	The chamfer / rounding command cannot be calculated.
		Check and adjust the chamfer / rounding command in the G code.

15.1.7 Tool compensation and tool related alarms (B100 - B1FF)

Alarm code	Name	Cause and corrective action
B100	Tool compen. interference TOOL COMPENSATION INTERFERENCE	Tool compensation is interfered or the calculation for the tool compensation coordinates is in error. Check and modify the programmed machining path or the tool radius for compensation.
B101	Cancel radius compen. in arc G40 EXECUTE IN ARC	This alarm occurs when you cancel the tool radius compensation when the block containing arc interpolation is executed. Modify the machining program to disable the tool radius compensation during linear interpolation.
B102	Enable radius compen. in arc G41/G42 EXECUTE IN ARC	This alarm occurs when you enable the tool compensation when the block containing arc interpolation is executed. Modify the machining program to disable the tool radius compensation during linear command.
B103	Radius interference ARC INTERF	The tool path is interfered after tool radius compensation. Check and modify the machining program or the tool radius for compensation.
B104	Tool compen. amount too small SHORT COMP LEN	The tool compensation path is too short. Check and modify the machining program or the tool radius for compensation.
B105	G41/G42 switch error ABNORMAL SWITCH G41/G42	An error occurs when the system switches the tool radius compensation to the right or left direction. Check and modify the machining program path.
B106	Use G31 in tool compen. EXECUTE G31 IN G41/G42	This alarm occurs when the system executes G31 during tool radius compensation. Check and modify the machining program.
B108	NURBS interpolation error NURBS INTERPOLATION ERROR	NURBS interpolation is in error. Check if the G code in the NURBS function complies with the command format. Check if the NURBS first control point is the same as the coordinate in the previous block.
B109	Insufficient 3D arc points 3D ARC INSUFFICIENT POINT	There is insufficient arc interpolation points. Check and modify the machining program.
B10B	Tool axis setting error ILLEGAL TOOL AXIS SETTING	This error occurs when Pr508 (Polygon cutting axis setting) is set to 0 and the system executes G51.2. This alarm occurs when Pr508 (Polygon cutting axis setting) is set to linear axis and the system executes G51.2. Correct the setting for Pr508 (Polygon cutting axis setting).
B10C	Illegal tool axis rotation mode ILLEGAL ROTATION MODE OF TOOL AXIS	This error occurs when you set Pr634 (Axis control variables - rotation axis feed mode) to linear mode for the tool axis and the system executes G51.2. Change the setting for Pr634 (Axis control variables - rotation axis feed mode) to rotation mode.
B10D	Illegal polygon interpolation G code ILLEGAL POLYGON INTERPOLATION GCODE	This alarm occurs when G51.2 (Polygon cutting) is enabled and you specified axial movement for the tool axis. Check the G code and modify the program.

15.1.8 Lathe related alarms (B300 - B3FF)

Alarm code	Name	Alarm cause and troubleshooting
B301	Threading pitch error THREADING PITCH ERR	The calculation result of the variable lead thread is less than 0. When you use the function of variable lead thread, if the lead increment per turn (K) is a negative value, the thread pitch becomes smaller with the increment of the number of turns. When the decrement in pitch is greater than the standard lead (F), this alarm occurs.
		Check the G code and modify the program.
B302	Spindle speed too fast SPINDLE SPD TOO HIGH	The turning feedrate for thread cutting is too fast.
		Reduce the spindle speed.
B303	Spindle/C axis switching Err INVALID SCAXIS CHANGE	You switch the system from Spindle mode to C axis mode while the C axis mode is disabled.
		Set the C axis to turning mode (Pr308 [Bit 15] = 0).
B304	Thread cutting prohibited in C axis mode THREADING UNDER CAXIS MODE	Thread cutting operation is prohibited in C axis mode.
		Switch the system from C axis mode to spindle mode to execute the thread cutting operation.
B305	C axis/spindle mode error SPINDLE MODE ERR	In C axis mode, both SP1 and SP2 are set to voltage mode.
		Adjust the spindle control mode settings of SP1 and SP2.
B306	Spindle mode breakpoint search error SEARCH CAXIS MODE ERR	In Spindle mode, the breakpoint search is in progress, but there is an M code for switching the system from Spindle mode to C axis mode.
		Reset the breakpoint search line number.
B307	C axis mode breakpoint search error SEARCH SAXIS MODE ERR	In C axis mode, the breakpoint search is in progress, but there is an M code for switching the system from C axis mode to Spindle mode.
		Reset the breakpoint search line number.
B308	Polar coordinate interpolation error POLAR INTERPOLATION ERR	An error occurs when the system executes the polar coordinate interpolation command.
		Check if the polar coordinate interpolation G code complies with the command format. Check if the polar coordinate interpolation supports G code command. Check if the plane selection is switched when the system is executing the polar coordinate command. Check if there is a specified tool number in the polar coordinate interpolation. The polar coordinate interpolation mode is only available in the following conditions: The system is in fixed spindle speed mode (G98). The tool nose compensation is canceled (G40). The system is in C axis mode (M2239 = 1).
B309	Spindle target speed not reached when cutting SPINDLE SPEED IS LESS THAN TARGET SPEED IN CUTTING	This error occurs when Pr51 [Bit 9] (Spindle target speed check during cutting) is enabled.
		M2256 (Spindle reaches the target speed) is off during cutting. Adjust Pr406, the tolerance for the spindle target speed.

15.1.9 Command related alarms (B600 - B6FF; B64x not included)

Alarm code	Name	Cause and corrective action
B600	G code error PPI TOKEN ERROR	G code error. Check the G code and modify the program.
B601	Excessive layers in subroutine call PPI BUF OVERFLOW	The subroutine called excessive program layers. The subroutine cannot call program layers exceeding 8 levels.
B602	No G code symbol PPI NO SYMBOL	No correct G code symbol. Check the G code and modify the program.
B603	Variable symbol error PPI INCORRECT VAR	The variable symbol is in error. Check the G code and modify the program.
B604	Illegal G code symbol PPI NONEXIST	The G code symbol is illegal. Check the G code and modify the program.
B605	No G code symbol PPI INCORRECT SYMBOL	There is no G code symbol. Check the G code and modify the program.
B606	Subroutine call error PPI SUBFUN NONEXIST	The subroutine call is in error. Modify the program.
B607	Subroutine file name error PPI SUBFUN NR ERR	The subroutine file name is in error. Modify the program.
B608	Subroutine layer No. error PPI SUBFUN OVERRANGE	The subroutine layer number is in error. Modify the program.
B609	Cycle EXEC, no homing PPI HOME NOT ALLOWED	G code is executed without homing. Execute homing for each axis first.
B60A	Dwell time cmd syntax error DWELL TIME MISMATCHED	The syntax for the dwell time command is in error. Check the G code and modify the program.
B60B	WP coord offset error OFF POS OVERLAPPED	An error occurs when the system computes the workpiece coordinate offset. Restart the controller or send it back for servicing.
B60C	Arc magnification ratio error PPI SCALE ERROR	The arc magnification ratio is in error. Check the G code and modify the program.
B60D	Homing midpoint Err PPI INTER POS NEXIST	An error occurs when the system performs homing through the intermediate point. Check the G code and modify the program.
B60E	Homing Err in Cyc machining HOME DUE TO CANNED CYCLE	Do not execute G28 / G29 / G30 during cyclic machining. Check the G code and modify the program.
B60F	G54 extension code err PPI G54 EXT ERROR	The specified range for the extended workpiece coordinates is in error. Check the G code specified range.
B610	Macro variable type error MACRO TYPE MISMATCH	The macro variable type is in error. Check the macro and modify the program.
B611	Macro not found MACRO NOT FOUND	The macro command is not found. Check the macro and modify the program.
B612	Macro line No. error MACRO INCORRECT LINE	The specified N is not found when the system executes the GO TO command. Modify the program.
B613	Macro bit setting error MACRO OUT OF RANGE	The setting for the macro bit is in error. Check the macro and modify the program.

Alarm code	Name	Cause and corrective action
B614	Macro divide by zero error MACRO DIVID ZERO	This error occurs when the macro performs division operation and the divisor is zero.
		Check the macro and modify the program.
B615	Macro command too long MACRO BUF OVERFLOW	The macro command is too long.
		Check the macro and modify the program.
B616	Macro Cmd operand not exist MACRO OPRND NOT EXIST	The macro command operand does not exist.
		Check the macro and modify the program.
B617	Macro command error MACRO ERROR	The macro command is in error.
		Check the macro and modify the program.
B618	Macro Cmd syntax error MACRO MISMATCH ERR	The macro syntax usage is illegal.
		Check the macro and modify the program.
B619	The macro operand syntax is in error MACRO BUF UNDERFLOW	The macro operand syntax is in error.
		Check the macro and modify the program.
B61A	Illegal macro command MACRO ILLEGAL	The macro command is illegal.
		Check the macro and modify the program.
B61B	GOTO tag is not found MACRO GOTO NOT FOUND	No corresponding tag is found in the G code.
		Check the G code and specify the correct tag.
B61C	No line No. given by GOTO MACRO GOTO FAIL	No corresponding line number is found in the G code.
		Check the G code and specify the correct line number.
B620	User-defined macro alarm MACRO USER ALRM	User-defined macro alarm is triggered.
		This user-defined macro alarm is triggered when G code specifies a non-zero value for #6000.
B621	Invalid halt code INVALID HALT CODE	The halt code is invalid.
		Check the G code and the halt code (M code) must be used individually.
B623	Feedrate is negative FEEDRATE IS NEGATIVE	The feedrate is set as a negative value.
		Check the G code and adjust the feedrate.
B625	Interpolation axis error INTERPOLATED AXES ERROR	Does not support linear axis and rotary axis synchronous interpolation.
		Use NC300 or above models.
B630	Excessive following error FOLLOW ERROR ALARM	The following error is too large.
		Make sure the servo parameter setting is correct. Check if the system parameter 643 is set too small.
B631	Hardware limit error HW LIMIT ERROR	The hardware limit is triggered.
		Move the axis out of the restricted range. Check the hardware limit wiring and polarity settings.
B632	1st software limit error SW LIMIT ERROR	The axis position exceeds the first software limit.
		Move the axis out of the restricted range.
B634	2nd software limit error SW LIMIT EXT ERROR	The axis position exceeds the second software limit.
		Move the axis out of the restricted range.
B636	Home sensor error ABNORMAL HOME SENSOR	The home sensor is in error.
		Check if the home sensor is installed correctly.
B637	MLC axis not stopped MLC AXIS IS NOT STOP	When the system switches between the MLC and NC axis modes, the MLC axis is still in motion.
		Modify the MLC program.

Alarm code	Name	Cause and corrective action
B638	1st software limit error (line No.) THE 1ST SOFTWARE LIMIT ERROR (LINE NUMBER)	This error occurs when Pr46 [Bit 15] (Pre-warning for software limit) is enabled.
		Modify the program. Check the parameter setting for the first software limit.
B639	2nd software limit error (line No.) THE 2ND SOFTWARE LIMIT ERROR (LINE NUMBER)	This alarm occurs when Pr46 [Bit 15] (Pre-warning for software limit) is enabled.
		Modify the program. Check the parameter setting of the second software limit. Exclude the factors that cause excessive following error: the servo gain is too low or motor power cable UVW connection error.
B650	Illegal G10 PAR definition ILLEGAL G10 PARAMETER	The G10 parameter definition is illegal.
		Check if the G code is used correctly.
B651	G10 PAR range error ILLEGAL PARAMETER RANGE	The G10 parameter range is set incorrectly.
		Check if the G code parameter range is correct.
B652	No spindle speed for cycle PPI CANNED S ERR	There is no spindle speed in the cycle command.
		Check the G code and specify the spindle speed.
B653	No feedrate for cycle PPI CANNED F ERR	There is no feedrate in the cycle command.
		Check the G code and specify the feedrate.
B654	Illegal cycle command PPI CANNED NOT EXIST	The usage of the cycle command is illegal.
		Check the G code and modify the cycle command.

15.1.10 Synchronous motion and temperature compensation related alarms (B640 - B645)

Alarm code	Name	Cause and corrective action
B640	Overheat OVERHEAT	The temperature exceeds the sensing range.
		Check the sensor's output specification.
B642	Temperature sensor disconnected THERMO SENSOR DISCONNECT	The temperature sensor is disconnected.
		Check if the wiring for the temperature sensor functions normally.
B643	Temperature detection error SENSOR WRONG	The temperature sensor is in error.
		Check if the temperature sensor is installed correctly.
B645	Excessive synchronous following error SYNCHRONOUS FOLLOW ERROR ALARM	This alarm occurs when the following error between the master and slave axes exceeds the range specified in Pr642 during synchronous motion control.
		Check if the servo gain for the master and slave axes is compatible. Eliminate the factors that cause the following error of the slave axis.

15.1.11 Lathe cycle command alarms (B10D, B6A1 - B6A5)

Alarm code	Name	Cause and corrective action
B10D	Polygon cutting G code error ILLEGAL POLYGON INTERPOLATION GCODE	This alarm occurs when G51.2 (Polygon cutting) is enabled and you have specified axis movement for the tool axis in the G code.
		Check and modify the machining program.
B6A1	No specified line No. in G70 - G73 G70 ~ G73 ILLEGAL LINE NUMBER	No specified line number is found in the cycle command.
		Check the G code and modify to the correct line number.
B6A2	No line No. given by G70 - G73 G70 ~ G73 NO LINE NUMBER	The cycle command did not specify the line number.
		Check the G code and add the correct line number.
B6A3	Cycle command taper error ILLEGAL TAPER AMOUNT	The taper calculation of the thread cutting cycle command is in error.
		Modify the G code and check related parameters.
B6A4	Chamfer command error GEOMETRY ERROR OF CHAMFER	The chamfer geometry dimension is incorrect.
		Modify the G code and check related parameters.
B6A5	Illegal drilling / tapping ILLEGAL DRILL / TAPPING	Drilling / tapping cannot be executed.
		Check the G code and the C axis status.

(This page is intentionally left blank.)

Revision History

Release Date	Version	Chapter	Revision contents
April, 2020	V1.0 (First edition)		

For relevant information about [Delta CNC Lathe Machine Solution - Operation and Maintenance Manual], please refer to:

(1) Delta CNC Lathe Machine Solution - G Command Guidelines

(2) Delta CNC NC Series Solution - MLC Application Manual

(This page is intentionally left blank.)