Smarter. Greener. Together.

Industrial Automation Headquarters

Delta Electronics, Inc.
Taoyuan Technology Center
18 Xinglong Road, Taoyuan District,
Taoyuan City 33068, Taiwan (R.O.C.)
TEL: 886-3-362-6301 / FAX: 886-3-371-6301

Asia

Delta Electronics (Jiangsu) Ltd.
Wujiang Plant 3
1688 Jiangxing East Road,
Wujiang Economic Development Zone
Wujiang City, Jiang Su Province, P.R.C. 215200
TEL: 86-512-6340-3008 / FAX: 86-769-6340-7290

Delta Greentech (China) Co., Ltd.
238 Min-Xia Road, Pudong District,
ShangHai, P.R.C. 201209
TEL: 86-21-58635678 / FAX: 86-21-58630003

Delta Electronics (Japan), Inc.
Tokyo Office
2-1-14 Minato-ku Shibadaimon,
Tokyo 105-0012, Japan
TEL: 81-3-5733-1111 / FAX: 81-3-5733-1211

Delta Electronics (Korea), Inc.
1511, Byucksan Digital Valley 6-cha, Gasan-dong,
Geumcheon-gu, Seoul, Korea, 153-704
TEL: 82-2-515-5303 / FAX: 82-2-515-5302

Delta Electronics Int'l (S) Pte Ltd.
4 Kaki Bukit Ave 1, #05-05, 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

Americas

Delta Products Corporation (USA)
Raleigh Office
P.O. Box 12173,5101 Davis Drive,
Research Triangle Park, NC 27709, U.S.A.
TEL: 1-919-767-3800 / FAX: 1-919-767-8080

Delta Greentech (Brasil) S.A.
Sao Paulo Office
Rua Itapeva, 26 - 3° andar Edificio Itapeva One-Bela Vista
01332-000-São Paulo-SP-Brazil
TEL: 55 11 3568-3855 / FAX: 55 11 3568-3865

Europe

Delta Electronics (Netherlands) B.V.
Eindhoven Office
De Witbogt 20, 5652 AG Eindhoven, The Netherlands
TEL: +31 (0)40-8003800 / FAX: +31 (0)40-8003898

Delta CNC Solution NC Series User Manual for Operation and Maintenance

Delta CNC Solution NC Series User Manual for Operation and Maintenance

www.deltaww.com

Smarter. Greener. Together.

Preface

Thank you for choosing this product. Before using the product, please read through this manual carefully in order to ensure the correct use of the product. In addition, please place the manual safely for quick reference whenever is needed.

This manual includes:

- Installation and inspection of NC controllers
- Wiring of NC controller pins
- Description of NC controller's function and operation
- Description of the system parameters
- Troubleshooting

Product features

- Built-in 32-bit high-speed dual CPU for multi-task execution and performance improvement
- Friendly HMI Interface
- Servo Gain Auto-tuning Interface for different mechanism specifications
- CNC Soft software tools to facilitate the development of customized screen images
- Front USB interface (port) to facilitate data access, data backup and parameters copy
- Different spindle control forms for the user to choose from: communication type or analog voltage type
- Serial I/O modules for flexible I/O configuration

How to use this manual:

This manual can be used as reference while studying NC controllers, which contains the information about the product installation, setting, as well as instructions of how to use and maintain this product. Before using and setting your NC controller, please read through this manual carefully.

DELTA technical services

Please consult the distributors or DELTA customer service center if any problem occurs.

Safety Precautions

- Please follow the instruction of pin assignment when wiring. Ground is a must.
- When the power is being supplied, do not disconnect the controller, change the wiring or touch the power source to avoid electric shock.

Please pay close attention to the following safety precautions during inspecting, installation, operating, maintenance and troubleshooting.

The symbols of “**DANGER**”, “**WARNING**” and “**STOP**” represent:

DANGER	**It indicates the potential hazards. It is possible to cause severe injury or fatal harm if not follow the instructions.**
WARNING	**It indicates the potential hazards. It is possible to cause minor injury or lead to serious damage of the product or even malfunction if not follow the instructions.**
STOP	**It indicates the absolute prohibited activity. It is possible to damage the product or cannot be used due to malfunction if not follow the instructions.**

Installation

DANGER

- Please follow the installation instructions in this manual; otherwise it may cause damage to the equipment.
- It is prohibited to expose the product to the environment containing water, corrosive gas, inflammable gas etc. Otherwise, electric shock or fire may occur.

Wiring

DANGER

- Please connect the ground terminal to class-3 ground system (under 100 Ω). Poor grounding may result in electric shock or fire.

Operation

WARNING

- Correctly plan out the I/O actions with MLC Editor Software, or abnormal results may occur.
- Before operation, please properly adjust the parameter settings of the machine, otherwise it may cause abnormal operation.
- Please ensure the emergency stop can be activated at any time, and avoid operating the machine in unprotected condition.

STOP

- Do not modify wiring while power is being supplied. Otherwise, it may cause personal injury due to electric shock.
- Never use a sharp-pointed object to touch the panel, as doing this might dent the screen and lead to malfunction of the controller.

Maintenance and Inspection

- While power is being supplied, do not disassemble the controller panel or touch the internal parts, otherwise electric shock may occur.
- Do not touch the ground terminal within 10 minutes after turning off the power, as the residual voltage may cause electric shock.
- Turn OFF the power first before replacing backup battery, and recheck the system settings afterwards.
- Do not block the vent holes during operation, as malfunction may easily occur due to poor ventilation.

Wiring Method

- Power supply: In order to avoid danger, use a 24 V_{DC} power supply for the controller and comply with the wire specification when wiring.
- Wiring materials: Use multi-stranded twisted-pair wires or multi-core shielded-pair wires to isolate all cables.
- The maximum cable length for remote I/O signals and DMCNET communication is 20 m and the maximum cable length for other signal cable is 10 m.
- To control the input and output signals, a 24 V_{DC} power is required for the controller I/O and remote I/O.

Wiring of Communication Circuit

- DMCNET wiring: The wiring materials should be in compliance with the standard specification.
- Please make sure the wiring between the controller and servo drive is tight and secure, as loose cables may cause abnormal operation.

If there is any difference of each version, please refer to DELTA's website for the latest information (http://www.delta.com.tw/industrialautomation/).

(This page is intentionally left blank.)

Table of Contents

4 Table of Group Menu

5 Mode Function

6 Position (POS) Group

7 Program (PRG) Group

13 Software Control Panel (SOFT) Group

14 MLC Special M and D Command

15 Troubleshooting

1 Inspection and Model Explanation

You can find product model explanation and introduction of each connector of NC controller in this chapter.

1.1 Inspection

In order to prevent the negligence during purchasing and delivery, please carefully inspect the items listed below:

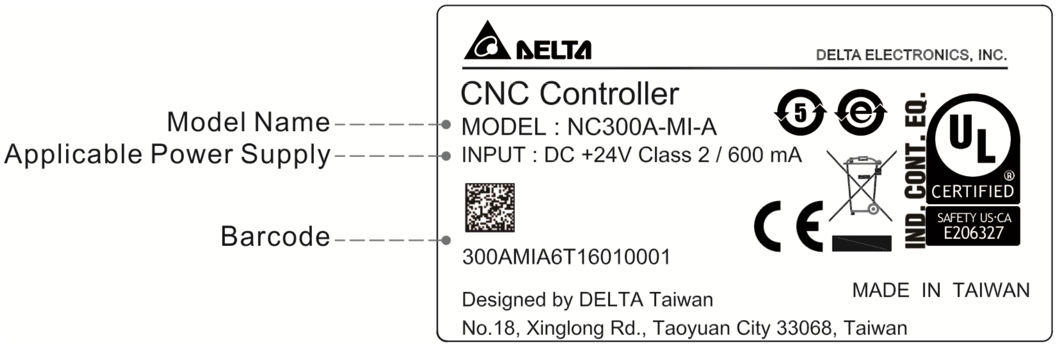
Check Item	Content
Check if the product is what you have purchased.	Check the model number specified on the controller's nameplate. Please refer to Model Explanation in section 1.2.
Check if keys and buttons can work properly	Press each button by hand. Normal keys can be pressed smoothly. [1]
Appearance	Visually check if there is any damage or scrape in the product's appearance.
Connector	Check if there is any loose or un-tightened connector.

If any of the above situations happens, please contact the distributors to solve the problems.

Note: Apart from NC _ _EM, this is applicable to other models.

1.2 Model Explanation

- Nameplate information



- Model Explanation

NC3_ _ / NC2_ _ series controller for milling system

NC300A	-	M	I	-	A
(1)		(2)	(3)		(4)

(1) Product Name

NC200: 3-axis 8-inch CNC controller

NC300: 3-axis 8-inch CNC controller

NC310: 3-axis 10-inch Horizontal CNC controller

NC311: 3-axis 10-inch Vertical CNC controller

(2) Application

M: For engraving and milling applications

(3) Type

I: All-in-one type (integrated with both primary and secondary panels)

S: Separated type (secondary panel not integrated)

(4) Version

A: Standard version

AE: English version

NC2_ _ series controller for lathe system

NC200A – L I – A

(1) (2) (3) (4)

(1) Product Name

NC200A: 2-axis 8-inch CNC controller (Standard)

NC200P: 2-axis 8-inch CNC controller (with built-in MPG)

(2) Application

L: For lathe applications

(3) Type

I: All-in-one type (integrated with both primary and secondary panels)

(4) Version

A: Standard version

AE: English version

AS: Simplified Chinese version

NC_ _EM series controller

NC 30 EM

(1) (2) (3)

(1) Product name

NC: Numeric controller

(2) Axis number

10: 2 axes

30: 4 axes

50: 6 axes

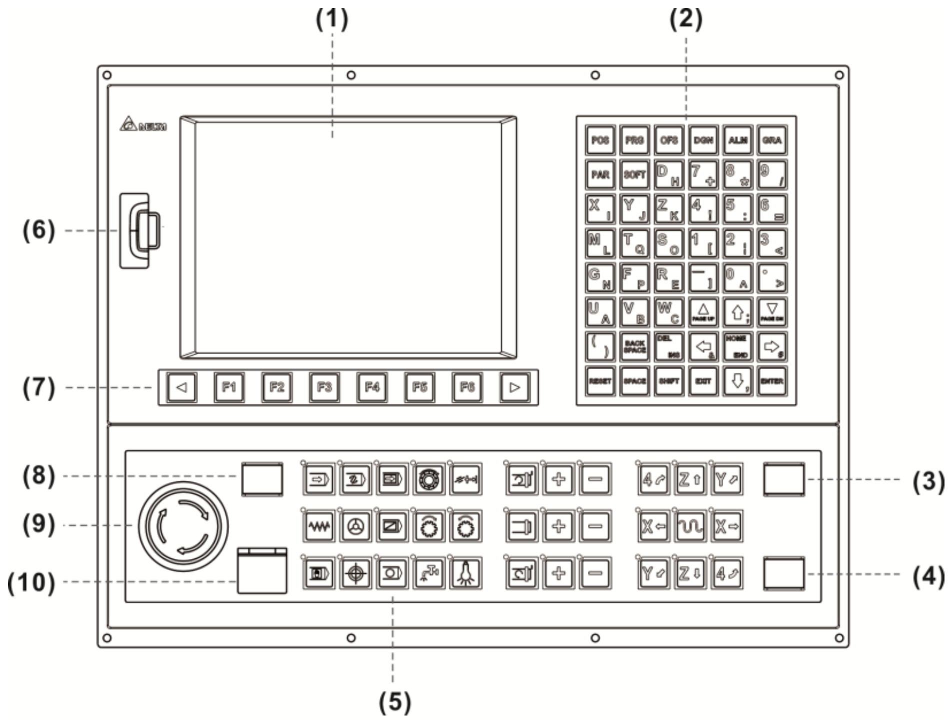
(3) Series

EM: EM series controller

PC: PC series controller

1.3 Description of each part

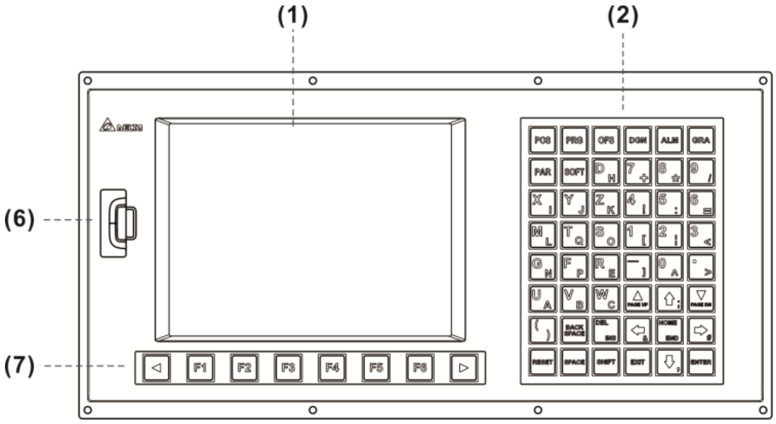
NC300A-MI-A_



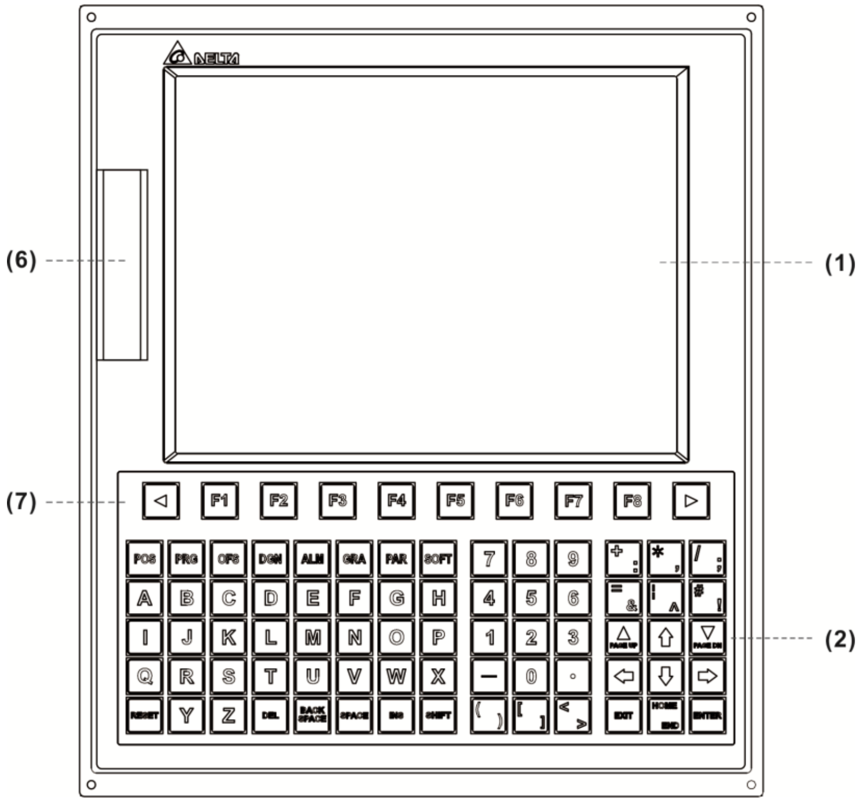
No.	Item	No.	Item
(1)	Screen	(6)	USB
(2)	The primary panel	(7)	Function keys
(3)	Cycle Start	(8)	Power ON
(4)	Feed Hold	(9)	Emergency Stop
(5)	The secondary panel	(10)	Power OFF

Note: In primary control panel, some keys have two characters. Directly press the key to input the upper character. For inputting the lower character, press the SHIFT key first.

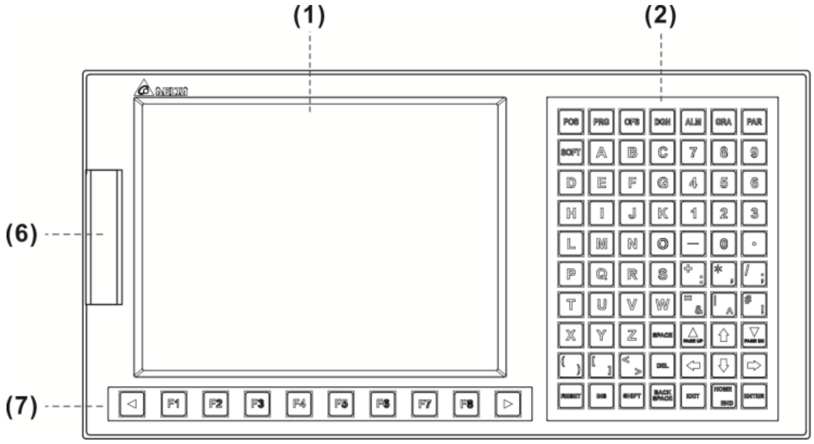
NC300A-MS-A_



NC311A-MS-A_

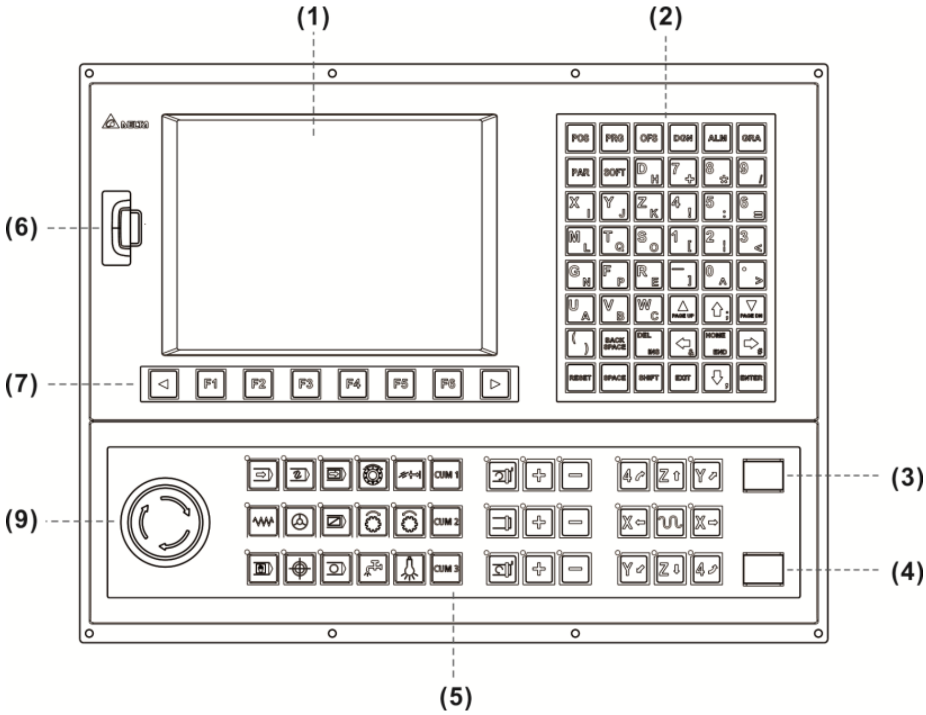


NC310A-MS-A_



No.	Item	No.	Item
(1)	Screen	(6)	USB
(2)	The primary panel	(7)	Function keys

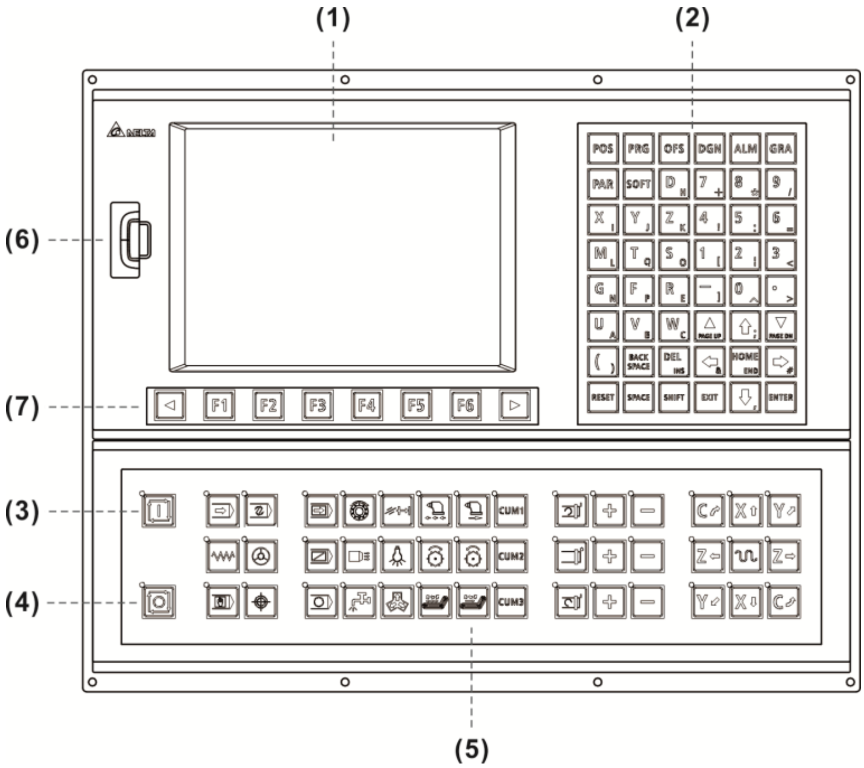
NC200A-MI-A_



No.	Item	No.	Item
(1)	Screen	(5)	The secondary panel
(2)	The primary panel	(6)	USB
(3)	Cycle Start	(7)	Function keys
(4)	Feed Hold	(9)	Emergency Stop

Note: In primary control panel, some keys have two characters. Directly press the key to input the upper character. For inputting the lower character, press the SHIFT key first.

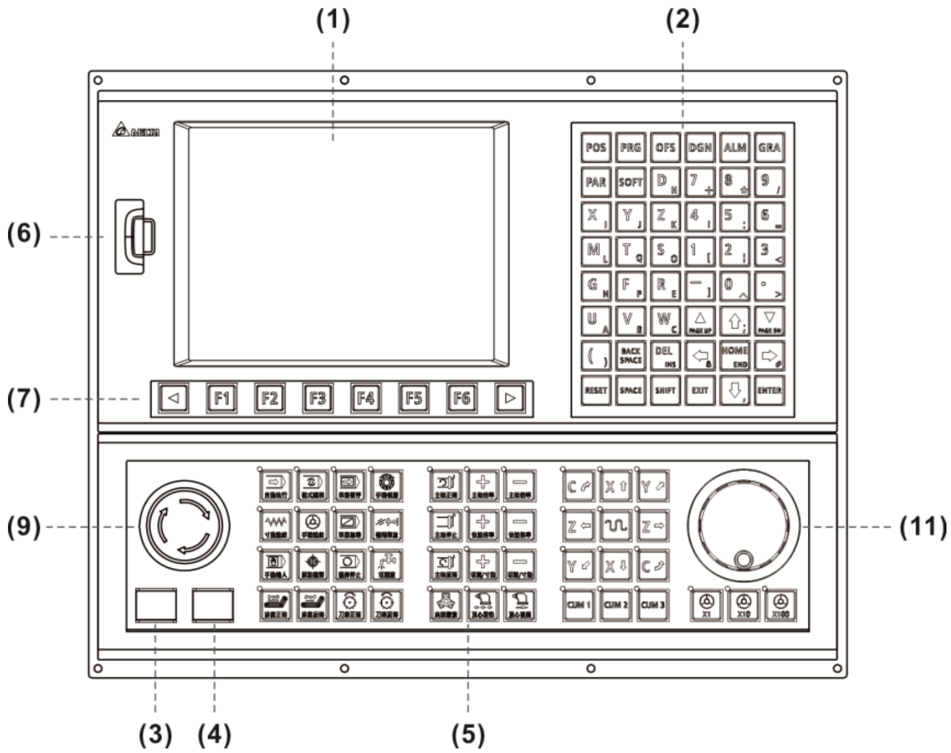
NC200A-LI-A_



No.	Item	No.	Item
(1)	Screen	(5)	The secondary panel
(2)	The primary panel	(6)	USB
(3)	Cycle Start	(7)	Function keys
(4)	Feed Hold	-	-

Note: In primary control panel, some keys have two characters. Directly press the key to input the upper character. For inputting the lower character, press the SHIFT key first.

NC200P-LI-A_

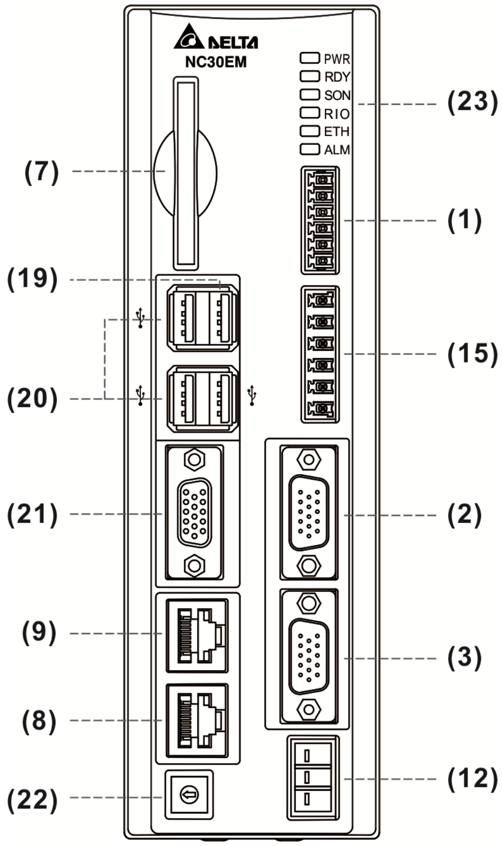


No.	Item	No.	Item
(1)	Screen	(6)	USB
(2)	The primary panel	(7)	Function keys
(3)	Cycle Start	(9)	Emergency Stop
(4)	Feed Hold	(11)	MPG
(5)	The secondary panel	-	-

Note: In primary control panel, some keys have two characters. Directly press the key to input the upper character. For inputting the lower character, press the SHIFT key first.

1

NC_ _EM



No.	Item	No.	Item
(1)	Remote I/O	(15)	HSI
(2)	MPG	(19)	The secondary panel
(3)	Spindle	(20)	USB (Connects to USB, keyboard and mouse)
(7)	CF card	(21)	VGA (connects to the screen)
(8)	DMCNET	(22)	Debugging mode
(9)	Ethernet	(23)	LED
(12)	24 VDC controller power	-	-

Installation

2

Please follow the instructions mentioned in safety precaution, ambient condition of storage and installation section for installing your controller. In addition, this chapter also provides the information of dimensions and specifications.

2

2.1 Safety precautions

Please follow the instructions mentioned in safety precautions, ambient condition of storage and installation section for installing NC controller. Otherwise, it might cause personnel injury or damage the equipment.

2.2 Ambient conditions of storage

Before installation, this product has to be kept in the shipping carton. If the product is temporarily not in use, please comply with the following instructions in order to retain the warranty coverage as well as for future maintenance:

- The product should be stored in dry and dust-free place.
- Store the product within an ambient temperature range of -20℃ to +60℃ (-4°F to 140°F).
- Store the product within a relative humidity range of 10% to 95%, non-condensing.
- Avoid storing the product in the environment of corrosive gas and liquid.
- The product should be installed in the environment without over-heat device, water drop, vapor, dust, oily dust, corrosive and inflammable gas, liquid, airborne dust, metal particles; and the environment should be solid without vibration or interference of electromagnetic noise.
- Specifications

NC3_ _ / NC2_ _ series controller for milling system

Model	NC300A-MI-A_	NC300A-MS-A_	NC311A-MS-A_	NC310A-MS-A_	NC200A-MI-A_
Working Environment	10% ~ 95% RH [0 ~ +55°C]				
Storage Environment	10% ~ 95% RH [-20 ~ +55°C]				
Cooling Method	Natural cooling				
Voltage	DC +24V (-10% ~ +15%) (built-in isolated circuit)				
Insulation Endurance	Between 24VDC and FG terminals: AC500V, 1 minute				
Power Consumption	15W (24V; 0.6A)				
Backup Battery	3V lithium battery CR2032 × 1				
Backup Battery Life	Varies with ambient temperature and working conditions; about 3 years in 25°C room temperature				
Dimensions (W) x (H) x (D) mm	400 x 309 x 123	400 x 200 x 97	290 x 335 x 95	435 x 221 x 95	400 x 309 x 123
Weight (kg)	4.1	3.1	3.8	3.8	3.8

NC2_ _ series controller for lathe system

Model	NC200A-LI-A_	NC200P-LI-A_
Working Environment	10% ~ 95% RH [0 ~ +55°C]	
Storage Environment	10% ~ 95% RH [-20 ~ +55°C]	
Cooling Method	Natural cooling	
Voltage	+24 VDC (-10% ~ +15%) (built-in isolated circuit)	
Insulation Endurance	Between 24 VDC and FG terminals: AC500V, 1 minute	
Power Consumption	15W (24V; 0.6A)	
Backup Battery	3V lithium battery CR2032 × 1	
Backup Battery Life	Varies with ambient temperature and working conditions; about 3 years in 25°C room temperature	
Dimensions (W) x (H) x (D) mm	400 x 320 x 91	400 x 320 x 130
Weight (kg)	4.5	4.7

NC_ _EM series controller

Model	NC_ _EM
Working Environment	10% ~ 95% RH [0 ~ +55°C]
Storage Environment	10% ~ 95% RH [-20 ~ +55°C]
Cooling Method	Natural cooling
Voltage	+24 VDC (-10% ~ +15%) (built-in isolated circuit)
Insulation Endurance	Between 24 VDC and FG terminals: AC500V, 1 minute
Power Consumption	15W (24V; 0.6A)
Backup Battery	3V lithium battery CR2032 × 1
Backup Battery Life	Varies with ambient temperature and working conditions; about 3 years in 25°C room temperature
Dimensions (W) x (H) x (D) mm	60 x 196 x 164
Weight (kg)	0.8

2.3 Ambient conditions of installation

Operation temperature: 0°C ~ 55°C (32°F ~ 131°F)
The ambient temperature of the controller for long-term reliability should be under 45°C (113°F).
Please place the product in a well-ventilated electric box when the temperature is over 45°C.
Also, pay attention to the vibration of the machine. Check if the vibration will influence the electronic device of the electric box.
Please observe the following precautions when selecting a mounting location. Failure to observe the following precautions may void the warranty.

- The product should be installed in the environment without over-heat device, water drop, vapor, dust, oily dust, corrosive and inflammable gas, liquid, airborne dust, metal particles; and the environment should be solid without vibration or interference of electromagnetic noise.
- Make sure the temperature and humidity of the installation site is within the range stipulated in the product specification.
- Store your NC controller in a place with vibration in a specified range.

2.4 Installation direction and space

NC series controller must be installed vertically on a dry and robust platform that is also NEMA standards-compliant. For good ventilation and cooling efficiency, sufficient clearance (50 mm or 2 inches) must be maintained between servo drive(s) and adjacent objects/partitions (walls).

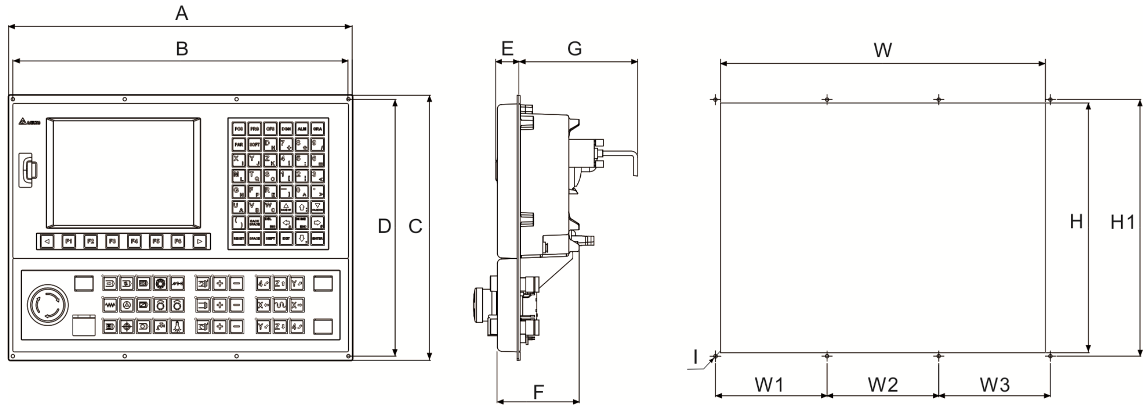
Note:

- For good ventilation and cooling efficiency, sufficient clearance must be maintained between adjacent objects and partitions (walls) surrounding the product. Otherwise, product failure may result.
- Do not block the ventilation slot of NC controller or product failure may result.

2.5 Dimensions

NC3_ _ series size marking

NC3_ _A-MI-A_/ NC2_ _A-MI-A_



NC3_ _A-MS-A_

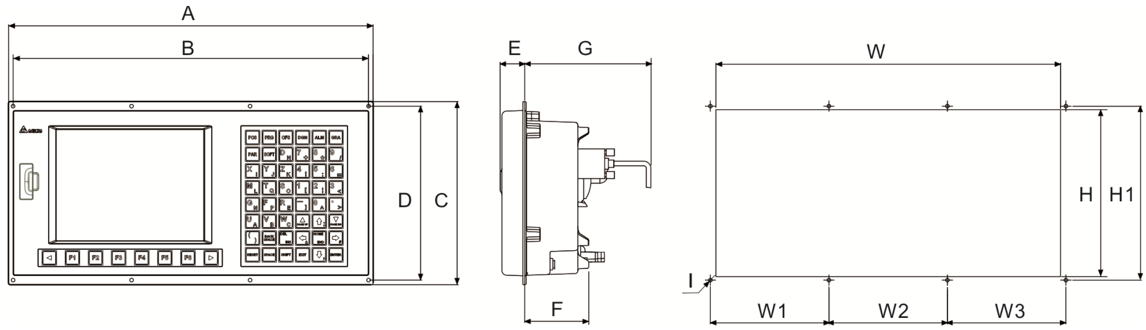


Table of Appearance Dimensions

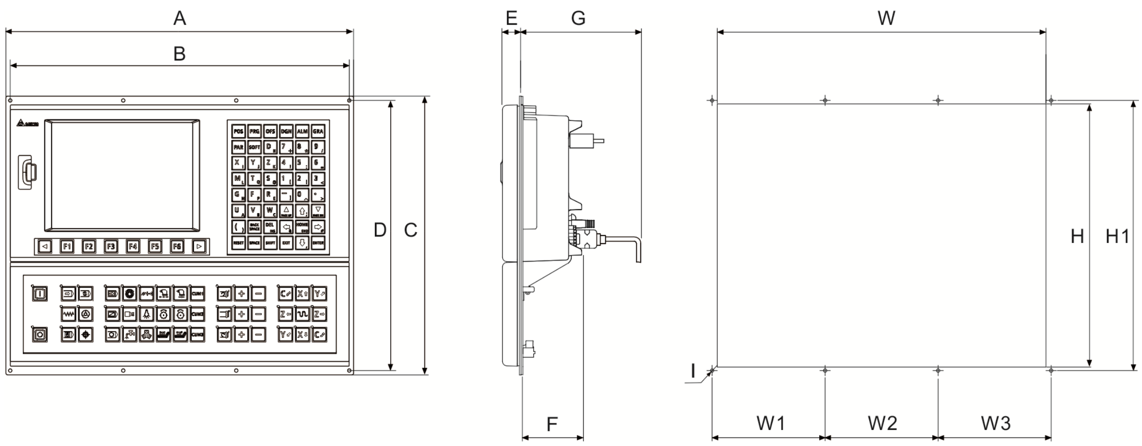
Model	NC300A-MI-A_ /NC200A-MI-A_	NC300A-MS-A_	NC311A-MS-A_	NC310A-MS-A_
A	400 mm	400 mm	290 mm	435 mm
B	390 mm	390 mm	280 mm	—
C	309 mm	200 mm	335.5 mm	221 mm
D	299 mm	190 mm	325.5 mm	—
E	25 mm	25 mm	23 mm	23 mm
F	70 mm	70 mm	70 mm	70 mm
G	130 mm(min)	130 mm(min)	130 mm(min)	130 mm(min)

Table of Cut-out Dimensions

Model	NC300A-MI-A_ /NC200A-MI-A_	NC300A-MS-A_	NC311A-MS-A_	NC310A-MS-A_
H	291±0.3 mm	182.5±0.3 mm	315.5±0.3 mm	191±0.3 mm
H1	299 mm	190 mm	325.5 mm	210.4 mm
W	378±0.3 mm	378±0.3 mm	270±0.3 mm	423±0.3 mm
W1	130 mm	130 mm	280 mm	141.47 mm
W2	130 mm	130 mm	__	141.47 mm
W3	130 mm	130 mm	__	141.47 mm
I	Φ4 mm	Φ4 mm	Φ4 mm	Φ4.5 mm

NC2__ series and NC__EM series size marking

NC2_ _A-LI-A_



NC2__P-LI-A_

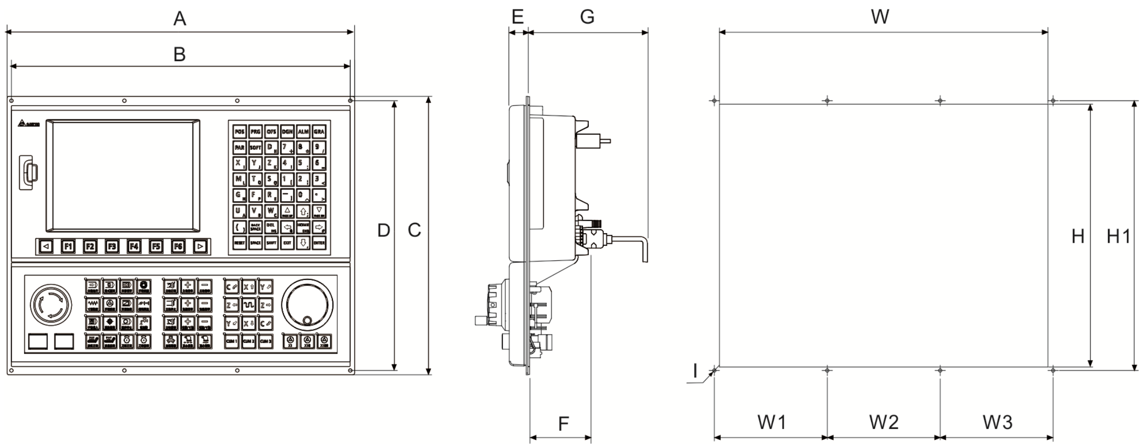


Table of Appearance Dimensions

Model	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(min)	130 mm(min)

Table of Cut-out Dimensions

Model	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

NC_ _EM

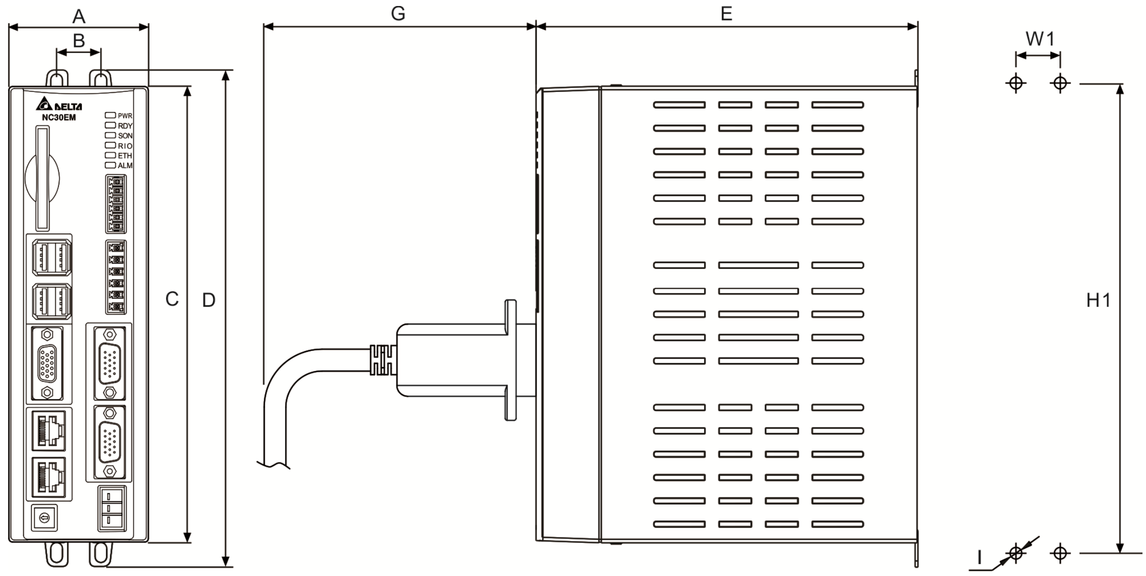


Table of Appearance Dimensions

Model / Size	NC_ _EM
A	60 mm
B	19 mm
C	196 mm
D	213 mm
E	164 mm
G	70 mm

Table of Cut-out Dimensions

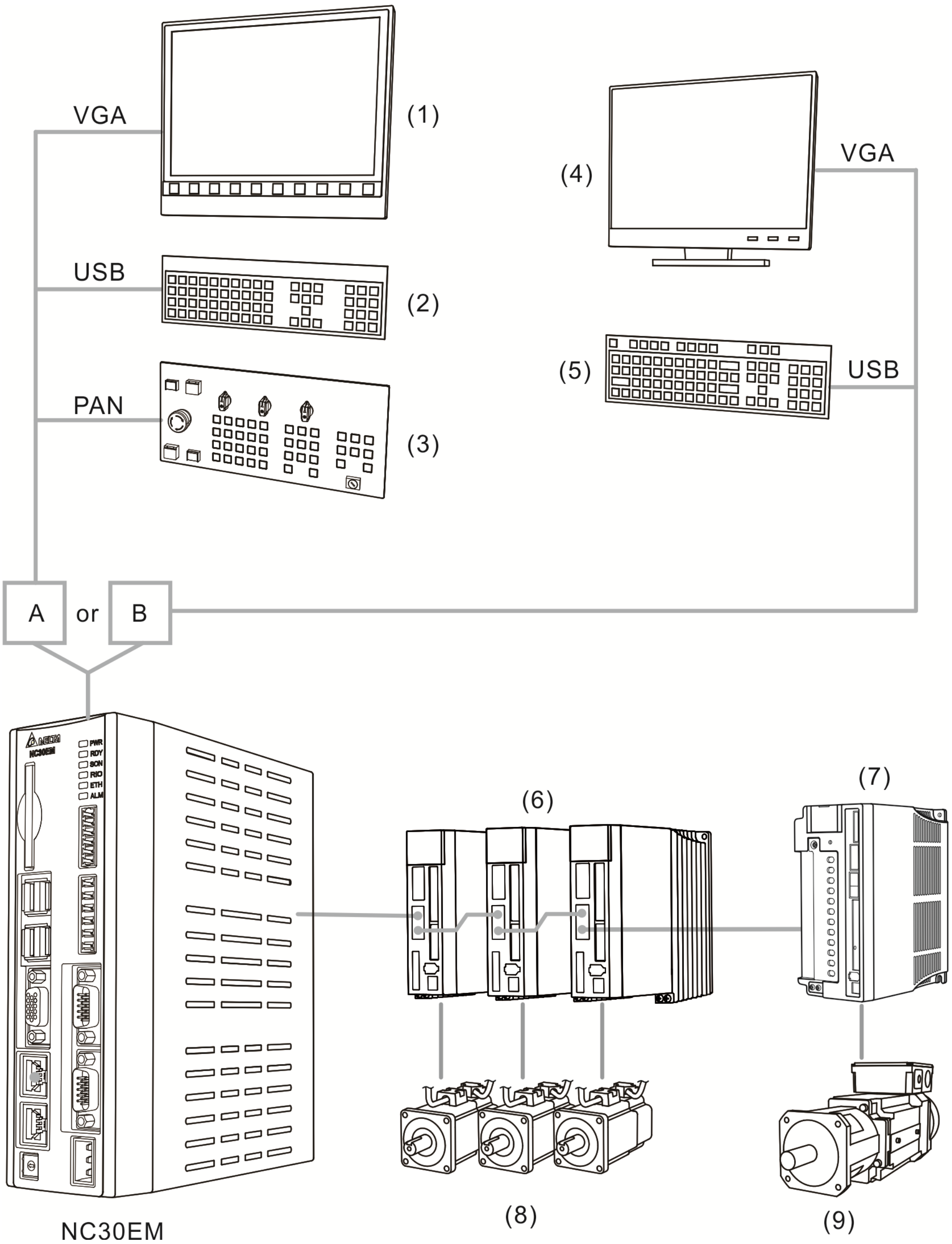
Cut-out	
H1	201±0.3 mm
W1	19±0.3 mm
I	M5

2.6 Panel installation (for NC_ _EM model)

- Installation method of controller panel

  A. Standard method: (1) Delta's panel; (2) Primary panel; (3) Secondary control panel.

  B. Non-standard method: (4) Commercial screen; (5) Commercial keyboard

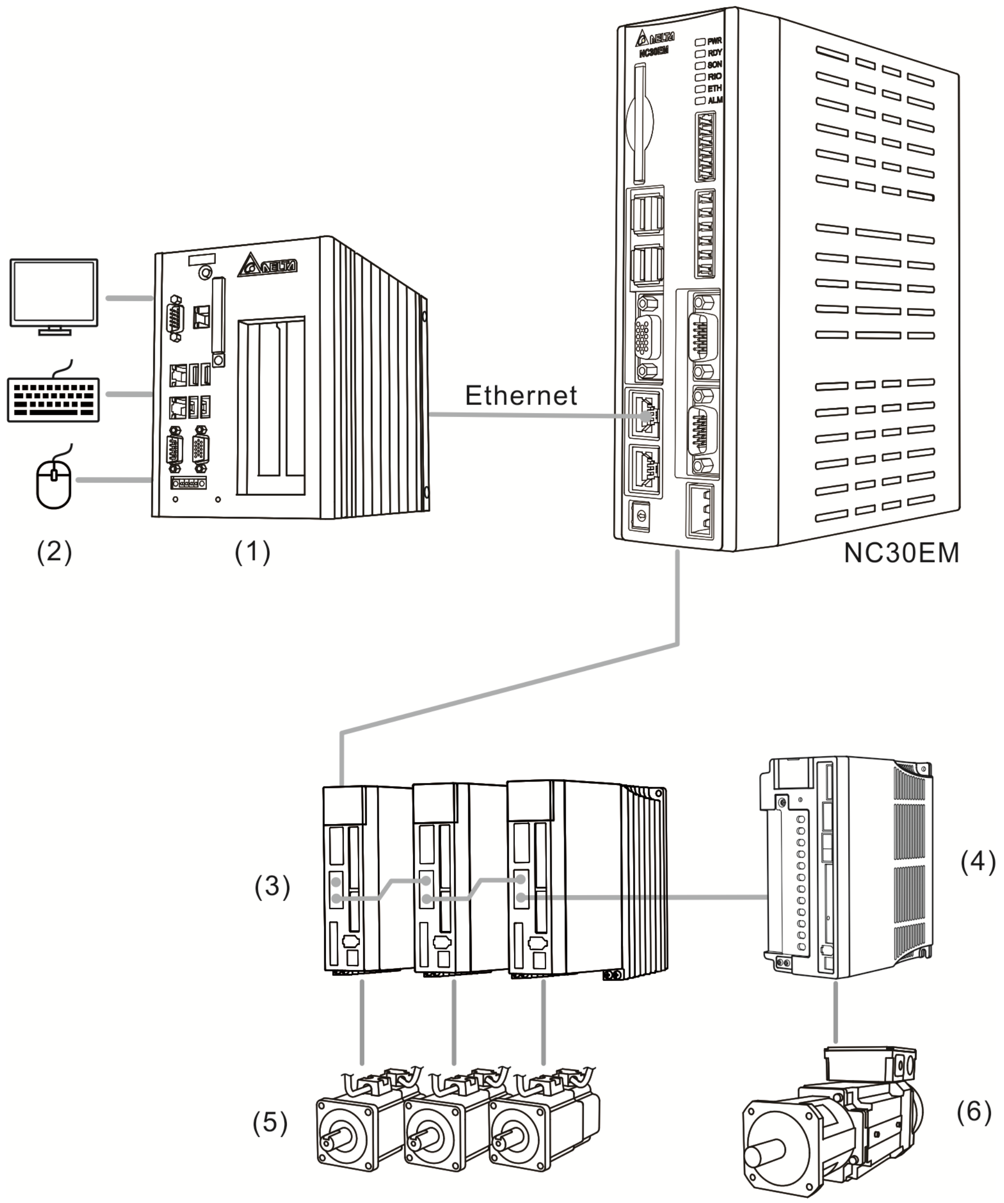


They all connect to (6) ASDA series AC servo drive(s), (7) Spindle servo drive(s) (ASDA-S), (8) ECMA series servo motor(s) and (9) ECMS series spindle motor(s).

2

- Through PC and Network

It includes (1) PC-based controller, (2) mouse and DELTA OpenCNC software, (3) ASDA series AC servo drive(s), (4) Spindle servo drive(s) (ASDA-S), (5) ECMA series servo motor(s) and (6) ECMS series spindle servo motor(s).



Wiring

3

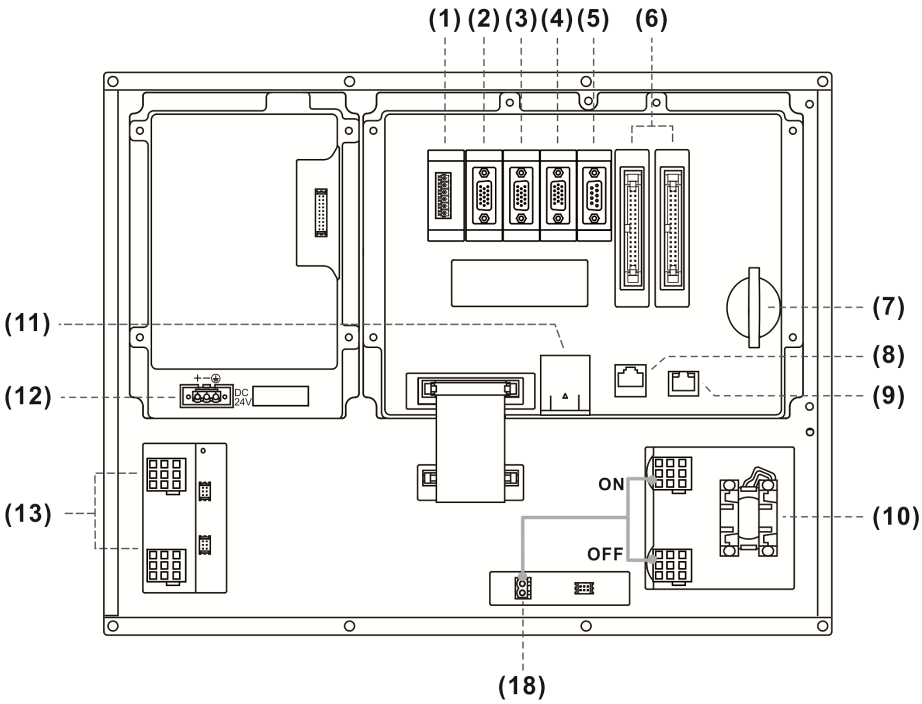
This chapter illustrates the wiring and connectors of NC controller.

3

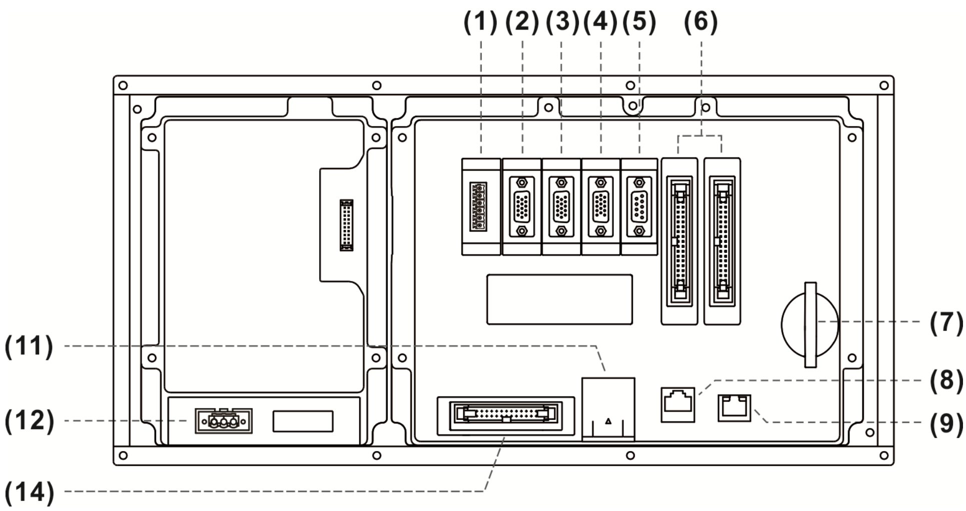
3.1 Wiring of system interface

■ NC3_ _ series

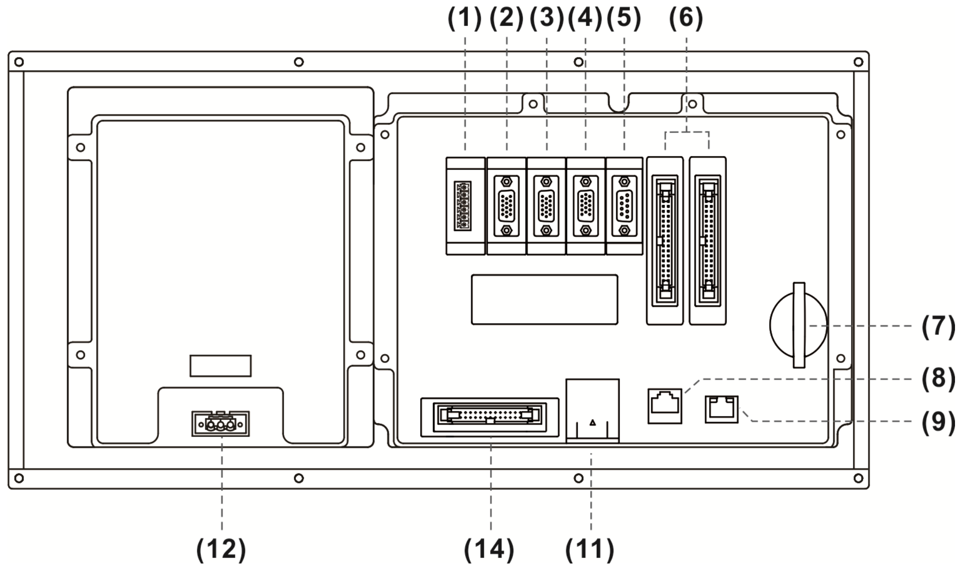
NC300A-MI-A_



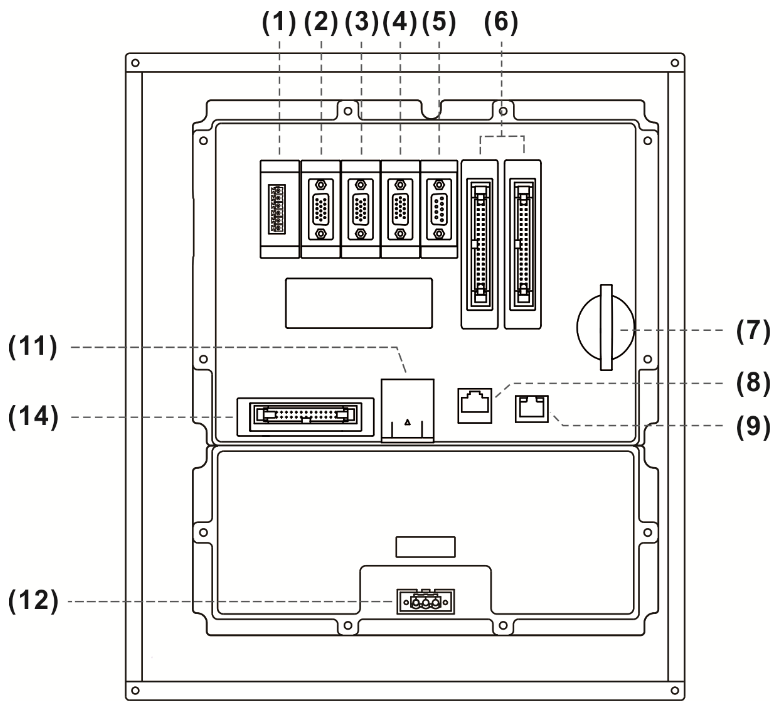
NC300A-MS-A_



NC310A-MS-A_



NC311A-MS-A_



Description of each connector:

No.	Connector	Description
(1)	Remote I/O	Each module features 32 pairs of inputs and outputs. Up to 20 M between stations and up to 160 M (20 x 8) of total length.
(2)	MPG	External MPG function with built-in 5 VDC power and 6 input points.

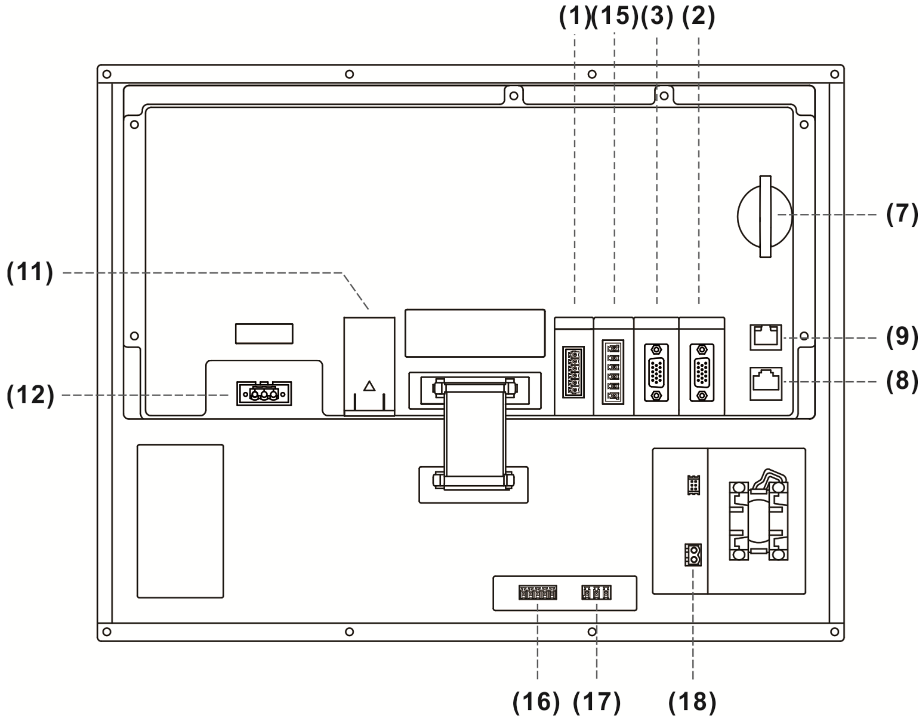
No.	Connector	Description
(3)	Spindle	One input point for spindle encoder signal.
		One output point for analog signal.
		G31 analog signal output
(4)	AXIS 1~4	Digital inputs of limits and origins of 4 axes
(5)	RS-485	For connecting to external devices via RS-485 communication
(6)	Local I/O	I/O 1: For connecting 16 digital inputs and 16 digital outputs I/O 2: For connecting 12 digital inputs and 12 digital outputs
(7)	CF Card	Storage for G Code program
(8)	DMCNET	For DMCNET communication
(9)	Ethernet	For DNC control and system monitoring
(10)	Emergency Stop	When Emergency Stop is pressed, the IES will be open-circuit and the system is stopped immediately.
(11)	Battery Holder	-
(12)	24 VDC Power	For 24 VDC power input
(13)	Cycle Start and Feed Hold keys	-
(14)	Wiring of the secondary panel	-
(18)	For secondary control panel	24 VDC power supply

Safety Precautions:

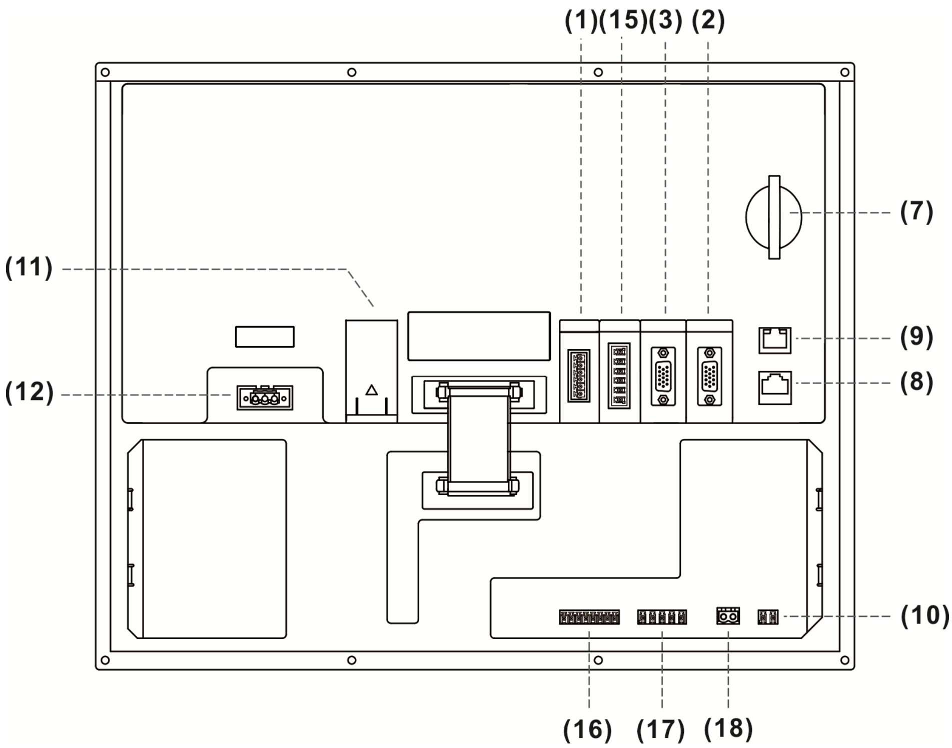
1. Ensure correct 24 VDC power connection.
2. On board and remote I/O require extra 24 VDC power to drive X input and Y output.
3. Short circuit the EMG (emergency stop) switch of the product to ready the controller.
4. For abnormal or emergency stop, disconnect the servo drive power by breaking the electromagnetic contactor (MC) with the Y output.

■ NC2_ _ series

NC200A-MI-A_

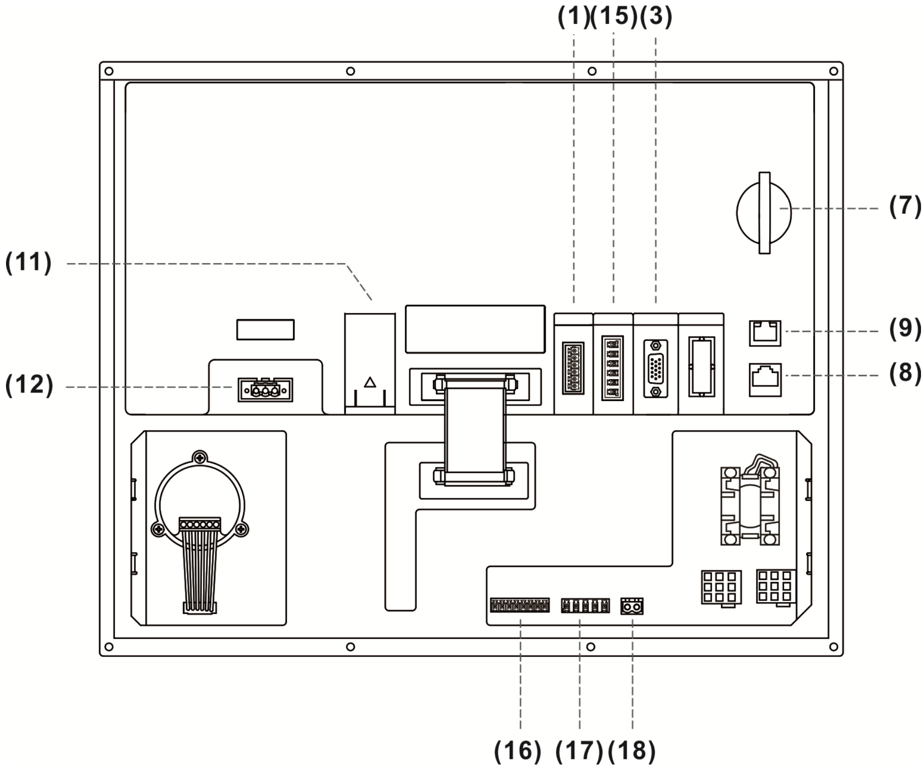


NC200A-LI-A_



3

NC200P-LI-A_



Description of each connector:

No.	Connector	Description	
(1)	Remote I/O	Each module features 32 pairs of inputs and outputs. Up to 20 M between stations and up to 160 M (20 x 8) of total length.	
(2)	MPG	External MPG function with built-in 5 VDC power, 7 input points and 1 output point.	
(3)	Spindle	One input point for spindle encoder signal.	
		One output point for analog signal.	
		One RS-485 serial communication terminal.	
(7)	CF Card	Storage for G Code program	
(8)	DMCNET	For DMCNET communication	
(9)	Ethernet	For DNC control and system monitoring	
(10)	Emergency Stop	-	
(11)	Battery Holder	-	
(12)	24 VDC Power	For 24 VDC power input	
(15)	HSI	The connector of G31 high speed and external emergency stop	
(16), (17)	Local I/O	MI series	DO: 3 digital outputs
			DI: 5 digital inputs
		LI series	DO: 5 digital outputs

No.	Connector	Description	
			DI: 8 digital inputs
(18)	24 VDC Power	(For the 24 VDC Power to the secondary panel)	

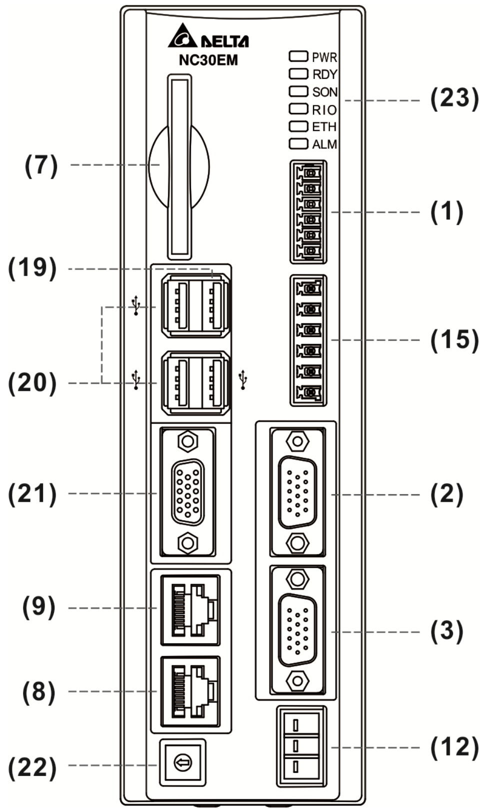
Safety Precautions

1. Ensure correct 24 VDC power connection.
2. Remote I/O requires extra power of 24 VDC to drive X input and Y output.
3. Short circuit the IES (emergency stop) switch of the product to ready the controller.
4. For abnormal or emergency stop, disconnect the servo drive power by breaking the electromagnetic contactor (MC) with the Y output.

■ NC_ _EM series

NC_ _EM

3



Description of each connector:

No.	Connector	Description
(1)	Remote I/O	Each module features 32 pairs of inputs and outputs. Up to 20 M between stations and up to 160 M (20 x 8) of total length.
(2)	MPG	External MPG function with built-in 5 V_{DC} power and 8 input points.
(3)	Spindle	One input point for spindle encoder signal.
		One output point for analog signal.
		One RS-485 serial communication terminal.
(7)	CF Card	Storage for G Code program
(8)	DMCNET	For DMCNET communication
(9)	Ethernet	For Ethernet communication
(12)	24 V_DC Power	For 24 V_{DC} power input
(15)	HSI	The connector of G31 high-speed and external emergency stop
(19)	PAN	One USB connector to connect the secondary control panel
(20)	USB	3 USB Host connectors to connect mouse, keyboard and flash drive
(21)	VGA	For connecting to the screen

No.	Connector	Description
(22)	MODE	For switching the debugging mode
(23)	LED	-

Debugging modes:

<table>
<tr><th>Setting value</th><th>Mode</th></tr>
<tr><td>0</td><td>Normal mode
LED
<table>
<tr><td>PWR</td><td>Green: System power on</td></tr>
<tr><td>RDY</td><td>Green flash: System ready
Orange: It is in debugging mode 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 OK</td></tr>
<tr><td>ETH</td><td>Green: Communication is OK</td></tr>
<tr><td>ALM</td><td>Red: System alarm occurs
Green flash: Software updating</td></tr>
</table></td></tr>
<tr><td>1</td><td>System updating</td></tr>
<tr><td>2</td><td>Default mode</td></tr>
<tr><td>3</td><td>Reset IP 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

1. Ensure correct 24 VDC power connection.
2. Remote I/O requires extra power of 24 VDC to drive X input and Y output.
3. For abnormal or emergency stop, disconnect the servo drive power by breaking the electromagnetic contactor (MC) with the Y output.

3

3.2 Connectors of NC controller

3.2.1 NC3_ _ series connectors

<table>
<tr><th>Connector</th><th>Function</th><th colspan="3">Description</th></tr>
<tr><td rowspan="5">0V, +24V
⏚</td><td rowspan="5">Power input for controller</td><td colspan="3">Connect the 24 VDC power (15 W at 0.6 A)</td></tr>
<tr><td>Terminal ID</td><td>Wire color</td><td>Description</td></tr>
<tr><td>+24V</td><td>Red</td><td>+24 VDC power</td></tr>
<tr><td>0V</td><td>White</td><td>0 VDC power</td></tr>
<tr><td>⏚</td><td>Green</td><td>Power grounding</td></tr>
<tr><td rowspan="7">I/O 1</td><td rowspan="7">On Board I/O 1</td><td colspan="3">DI/DO range (X0 ~ X15, Y0 ~ Y15)</td></tr>
<tr><td>PIN ID</td><td colspan="2">Description</td></tr>
<tr><td>P1 ~ P16</td><td colspan="2">Digital inputs for X0 ~ X15, 16 points in total (8 ~ 25 mA)</td></tr>
<tr><td>P19 ~ P34</td><td colspan="2">Digital outputs for Y0 ~ Y15, up to 16 points in total (< 120 mA)</td></tr>
<tr><td>P18</td><td colspan="2">Digital inputs COM, which can be connected to +24 VDC or 0 V</td></tr>
<tr><td>P17</td><td colspan="2">Digital output COM, which is connected to 0 V</td></tr>
<tr><td colspan="3">Power supply specifications: voltage: < 24 VDC; current < 60 mA</td></tr>
<tr><td rowspan="7">I/O 2</td><td rowspan="7">On Board I/O 2</td><td colspan="3">DI/DO range: (X16 ~ X27, Y16 ~ Y27)</td></tr>
<tr><td>PIN ID</td><td colspan="2">Description</td></tr>
<tr><td>P1 ~ P12</td><td colspan="2">Digital inputs for X16 ~ X27, 12 points in total (8 ~ 25 mA)</td></tr>
<tr><td>P19 ~ P30</td><td colspan="2">Digital outputs for Y16 ~ Y27, 12 points in total (< 120 mA)</td></tr>
<tr><td>P18</td><td colspan="2">Digital inputs COM, which can be connected to +24 VDC or 0 V</td></tr>
<tr><td>P17</td><td colspan="2">Digital output COM, which is connected to 0 V</td></tr>
<tr><td colspan="3">Power supply specifications: voltage: < 24 VDC; current < 60 mA</td></tr>
<tr><td rowspan="4">AXIS 1 ~ 4</td><td rowspan="4">Limits and home sensor of each axis</td><td colspan="3">Axis 1 ~ Axis 4, the positive/negative limits and home sensor input terminals, up to 12 points (Operating current: 8 ~ 25 mA)</td></tr>
<tr><td>PIN ID</td><td colspan="2">Description</td></tr>
<tr><td>P1 ~ P3</td><td colspan="2">Input points for positive limit, negative limit, and home point of Axis 1
(Special M [M2144], [M2145], [M2146])</td></tr>
<tr><td>P4 ~ P6</td><td colspan="2">Input points for positive limit, negative limit, and home point of Axis 2
(Special M [M2148], [M2149], [M2150])</td></tr>
</table>

Connector	Function	Description	
		P7 ~ P9	Input points for positive limit, negative limit, and home point of Axis 3 (Special M [M2152], [M2153], [M2154])
		P10 ~ P12	Input points for positive limit, negative limit, and home point of Axis 4 (Special M [M2156], [M2157], [M2158])
		P13 ~ P15	Input COM, which can be connected to +24 VDC or 0V
SPINDLE	Spindle specific connector	Including feedback of spindle, analog output and 2 high-speed inputs	
		PIN ID	Description
		P1	HIS_COM, for +24 VDC or 0V
		P2	HIS_1 (Counter C78, Input [M2142])
		P3	HIS_2 (Counter C79, Input [M2143])
		P4	SP_OUT
		P5	SP_GND
		P6	EMG_GND
		P7	EMG_IN
		P8	SP_A+
		P9	SP_A-
		P10	DC +5V_OUT
		P11	SP_B+
		P12	SP_B-
		P13	SP_Z+
		P14	SP_Z-
		P15	GND
MPG	MPG connection	6 DI points and one MPG differential input terminal.	
		PIN ID	Description
		P1	DI_COM, for +24 VDC or 0 V
		P2 ~ P7	DI (X28 ~ X33)
		P8	Reserved
		P9	Reserved
		P10	DC +5V_OUT(< 300 mA)
		P11	XA+
		P12	XA-
		P13	XB+
		P14	XB-
		P15	GND

<table>
<tr><th>Connector</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 serial connection</td><td colspan="2">Remote I/O module communication terminal, which ranges from X256 ~ X511 and Y256 ~ Y511. Every remote module has 32 inputs and 32 outputs and may connect up to 8 modules.</td></tr>
<tr><td>PIN ID</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 connection</td><td colspan="2">Connect PC with RJ45 connector with normal network cable.</td></tr>
<tr><td>PIN ID/color at end A</td><td>PIN ID/color at 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 communication</td><td colspan="2">Connect to Delta's servo drive of DMCNET type with standard RJ45 connector. Wiring method is the same as wiring for ETHERNET connector.</td></tr>
<tr><td>EMG</td><td>Emergency stop switch</td><td colspan="2">Press to open the circuit.</td></tr>
<tr><td>Power On</td><td>Power On contact</td><td colspan="2">Power On contact</td></tr>
<tr><td>Power Off</td><td>Power Off contact</td><td colspan="2">Power Off contact</td></tr>
<tr><td>IES</td><td>EMG emergency stop contact</td><td colspan="2">It controls the emergency stop contact and activates the EMG flag in broken circuit. (The EMG stop button is wired to IES terminal block.)</td></tr>
<tr><td>RS-485</td><td>RS-485 serial communication port</td><td colspan="2">Connect to external devices via RS-485 serial communication.</td></tr>
</table>

3

3.2.2 NC2_ _ series connectors

<table>
<tr><th>Terminal ID</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">Connect the 24 VDC power (15 W at 0.6 A)</td></tr>
<tr><td>Terminal ID</td><td>Description</td></tr>
<tr><td>+24V</td><td>+24 VDC power</td></tr>
<tr><td>0V</td><td>0 VDC power</td></tr>
<tr><td>⏚</td><td>Power grounding</td></tr>
<tr><td rowspan="4">0V, +24V</td><td rowspan="4">Power input for secondary control panel.</td><td colspan="2">Connect the 24 VDC power (15 W at 0.6 A)</td></tr>
<tr><td>Terminal ID</td><td>Description</td></tr>
<tr><td>+24V</td><td>+24 VDC power</td></tr>
<tr><td>0V</td><td>0 VDC power</td></tr>
<tr><td rowspan="12">I/O</td><td rowspan="6">DI</td><td colspan="2">DI terminal; power supply specification: voltage < 24 VDC; current 8 ~ 25 mA
MI series</td></tr>
<tr><td>PIN ID</td><td>Description</td></tr>
<tr><td>X112 ~ X116</td><td>Digital inputs for X112 ~ X116, 5 points in total</td></tr>
<tr><td colspan="2">LI series</td></tr>
<tr><td>PIN ID</td><td>Description</td></tr>
<tr><td>X112 ~ X119</td><td>Digital inputs for X112 ~ X119, 8 points in total</td></tr>
<tr><td rowspan="6">DO</td><td colspan="2">DO terminal; power supply specification: voltage < 24 VDC; current < 120 mA
MI series</td></tr>
<tr><td>PIN ID</td><td>Description</td></tr>
<tr><td>Y112 ~ Y114</td><td>Digital outputs for X112 ~ X114, 3 points in total</td></tr>
<tr><td colspan="2">LI series</td></tr>
<tr><td>PIN ID</td><td>Description</td></tr>
<tr><td>Y112 ~ Y116</td><td>Digital outputs for X112 ~ X116, 5 points in total</td></tr>
<tr><td rowspan="10">SPINDLE</td><td rowspan="10">Spindle specific connector</td><td colspan="2">Including feedback of spindle, analog output and 1 terminal of RS-485 serial communication</td></tr>
<tr><td>PIN ID</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>EMG_IN, for+24 VDC</td></tr>
<tr><td>P8</td><td>SP_A+</td></tr>
</table>

Terminal ID	Function	Description	
		P9	SP_A-
		P10	DC +5V_OUT
		P11	SP_B+
		P12	SP_B-
		P13	SP_Z+
		P14	SP_Z-
		P15	GND
MPG	MPG connection	7 DI points, 1 DO point and one MPG differential input terminal.	
		PIN ID	Description
		P1	DI_COM, for +24 VDC or 0 V
		P2 ~ P7	DI (X28 ~ X33)
		P8	DI (X26)
		P9	DO (Y27)
		P10	DC +5V_OUT(< 300 mA)
		P11	XA+
		P12	XA-
		P13	XB+
		P14	XB-
		P15	GND
HSI	HSI and EMG terminal	2 HSI input terminals and 1 EMG terminal	
		PIN ID	Description
		P1	EMG_IN
		P2	EMG_GND
		P3	HIS_1 (Counter C78, Input [M2142])
		P4	HSI_COM, for +24 VDC or 0 V
		P5	HIS_2 (Counter C79, Input [M2143])
		P6	HSI_COM, short-circuits with P4
REMOTE I/O	Remote I/O module serial connection	Remote I/O module communication terminal, which ranges from X256 ~ X511 and Y256 ~ Y511. Every remote module has 32 inputs and 32 outputs and may connect up to 8 modules.	
		PIN ID	Description
		P1	TX+
		P2	TX-
		P3	RX-
		P4	RX+
		P5	GND
		P6	SHIELD

<table>
<tr><th>Terminal ID</th><th>Function</th><th colspan="2">Description</th></tr>
<tr><td rowspan="10">ETHERNET</td><td rowspan="10">Ethernet connection</td><td colspan="2">Connect PC with RJ45 connector with normal network cable. See below for wire jumping.</td></tr>
<tr><td>PIN ID/color at end A</td><td>PIN ID/color at 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 connection</td><td colspan="2">Connect to Delta's servo drive of DMCNET type with standard RJ45 connector. Wiring method is the same as wiring for ETHERNET connector.</td></tr>
<tr><td>EMG</td><td>Emergency stop switch</td><td colspan="2">Press to open the circuit.</td></tr>
<tr><td>Power On</td><td>Power On contact</td><td colspan="2">Power On contact</td></tr>
<tr><td>Power Off</td><td>Power Off contact</td><td colspan="2">Power Off contact</td></tr>
<tr><td>IES</td><td>EMG emergency stop contact</td><td colspan="2">It controls the emergency stop contact and activates the EMG flag in broken circuit. (The EMG stop button is wired to IES terminal block.)</td></tr>
</table>

Note:

1. The IES connector is the input for the EMG emergency stop. Enable EMG flag when breaking circuit.
2. The key light indicators of Power On/ Power Off require 24 V_{DC} power supply.

3.2.3 NC_ _EM series connectors

Terminal ID	Function	Description	
0V, +24V ⏚	Power input for controller	Connect the 24 VDC power (15 W at 0.6 A)	
		Terminal ID	Description
		+24V	+24 VDC power
		0V	0 VDC power
		⏚	Power grounding
SPINDLE	Spindle specific connector	Including feedback of spindle, analog output and 1 terminal of RS-485 serial communication	
		PIN ID	Description
		P1	RS485_GND
		P2	RS485_D+
		P3	RS485_D-
		P4	SP_OUT
		P5	SP_GND
		P6	Reserved
		P7	Reserved
		P8	SP_A+
		P9	SP_A-
		P10	DC +5V_OUT
		P11	SP_B+
		P12	SP_B-
		P13	SP_Z+
		P14	SP_Z-
		P15	GND
MPG	MPG connection	8 DI points and one MPG differential input terminal.	
		PIN ID	Description
		P1	DI_COM, for +24 VDC or 0 V
		P2 ~ P9	DI (X0 ~ X7)
		P10	DC +5V_OUT(< 300 mA)
		P11	XA+
		P12	XA-
		P13	XB+
		P14	XB-
		P15	GND
HSI	HSI and EMG terminal	2 HSI input terminals and 1 EMG terminal	
		PIN ID	Description
		P1	EMG_IN (+ 5 VDC output)

<table>
<tr><th>Terminal ID</th><th>Function</th><th colspan="2">Description</th></tr>
<tr><td rowspan="5"></td><td rowspan="5"></td><td>P2</td><td>MODE_ENABLE (+ 5 VDC output)</td></tr>
<tr><td>P3</td><td>HIS_1 (Counter C78, Input [M2142])</td></tr>
<tr><td>P4</td><td>GND</td></tr>
<tr><td>P5</td><td>HIS_2 (Counter C79, Input [M2143])</td></tr>
<tr><td>P6</td><td>HSI_COM, for +24 VDC or 0 V</td></tr>
<tr><td rowspan="8">REMOTE I/O</td><td rowspan="8">Remote I/O module serial connection</td><td colspan="2">Remote I/O module communication terminal, which ranges from X256 ~ X511 and Y256 ~ Y511. Every remote module has 32 inputs and 32 outputs and may connect up to 8 modules.</td></tr>
<tr><td>PIN ID</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>Terminal for the secondary panel</td><td colspan="2">It is for connecting secondary control panel, which cannot connect to USB device.</td></tr>
<tr><td>USB</td><td>USB port</td><td colspan="2">This is for connecting USB devices, such as flash drive, mouse and keyboard. Serial connection is not supported.</td></tr>
<tr><td>VGA</td><td>VGA connector</td><td colspan="2">It can output to external monitor (60 Hz only).</td></tr>
<tr><td rowspan="10">ETHERNET</td><td rowspan="10">Ethernet connection</td><td colspan="2">Connect PC with RJ45 connector with normal network cable. See below for wire jumping.</td></tr>
<tr><td>PIN ID/color at end A</td><td>PIN ID/color at 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 communication</td><td colspan="2">Connect to Delta's servo drive of DMCNET type with standard RJ45 connector. Wiring method is the same as wiring for ETHERNET connector.</td></tr>
<tr><td>MODE</td><td>Debugging mode</td><td colspan="2">For switching the debugging mode.</td></tr>
</table>

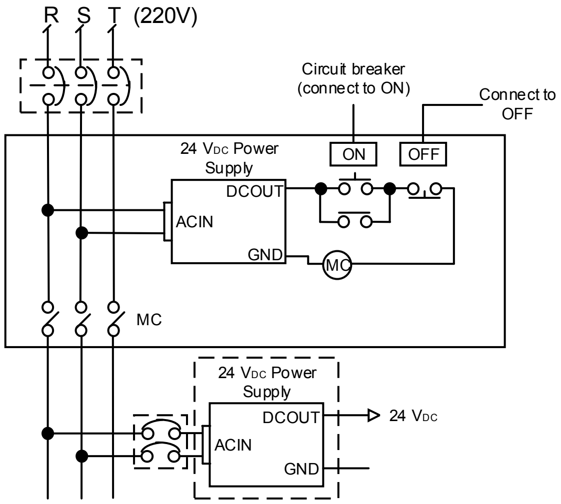
3

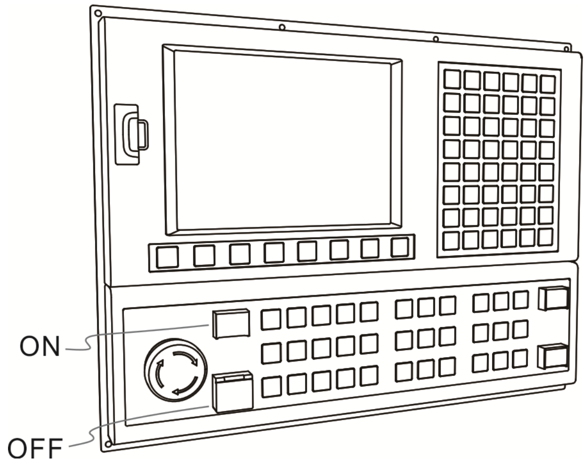
3.3 Power wiring

NC series controller has two kinds of power wiring, switch control and directly power supply.

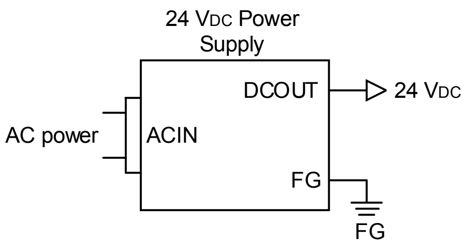
- Switch control

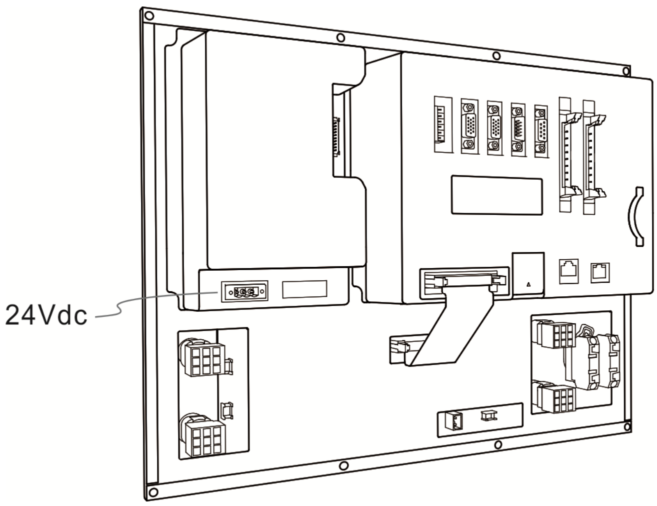
  See the figure below. Power On is NO contact and Power Off is NC contact. MC is the coil of magnetic contactor which is the device of self-contained power supply.





- Directly power supply

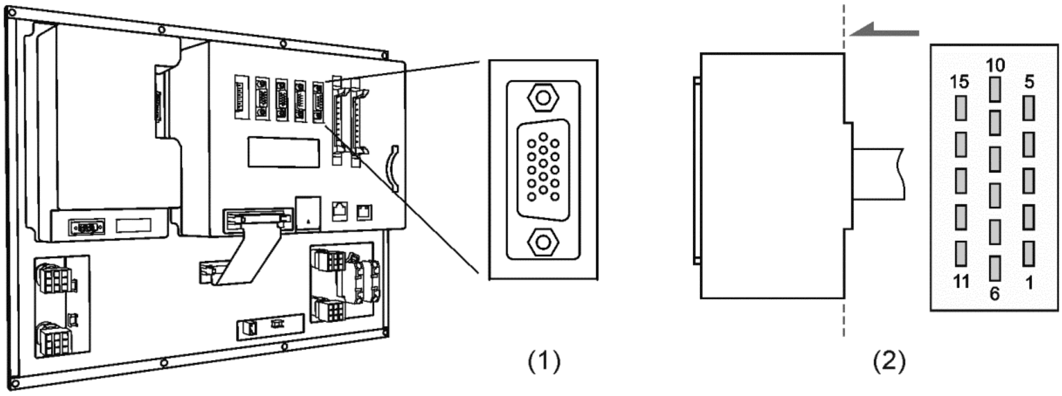




3.4 Wiring for RS-485 connector

NC controller provides one RS-485 serial communication connector for external devices.

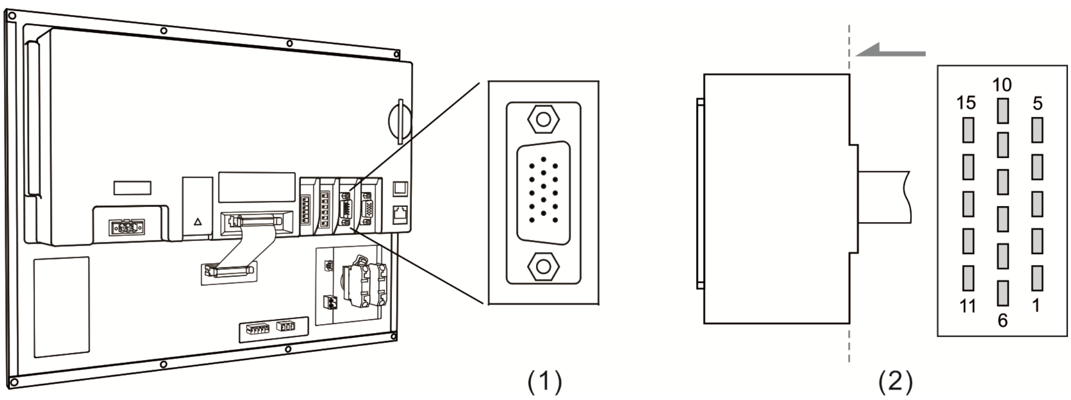
See below for NC3_ _ series PIN assignment.



(1) RS-485 connector on NC controller (Female); (2) RS-485 connector (Male)

Model	Terminal ID	Pin No	Function
NC3_ _	RS-485	PIN 1	D+
		PIN 6	D-

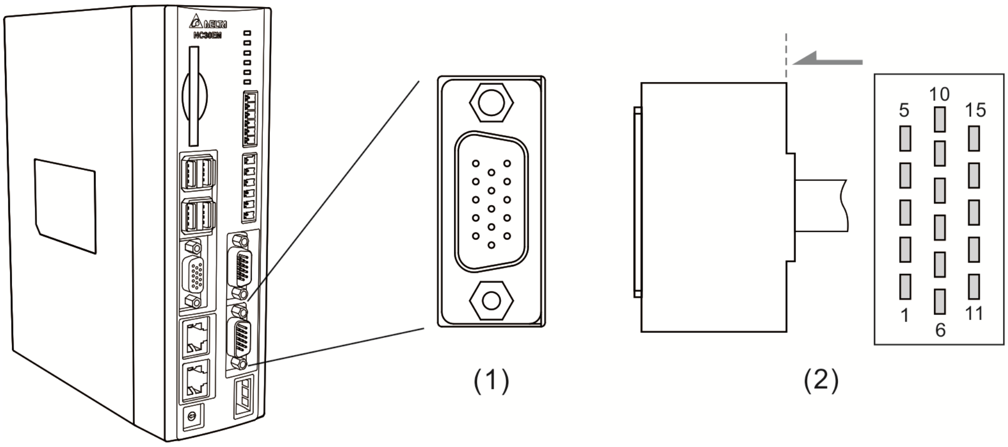
See below for NC2_ _ series PIN assignment.



(1) RS-485 connector on NC controller (Female); (2) RS-485 connector (Male)

Model	Terminal ID	Pin No	Function
NC2_ _	SPINDLE	PIN 2	D+
		PIN 3	D-

See below for NC_ _ EM series PIN assignment.



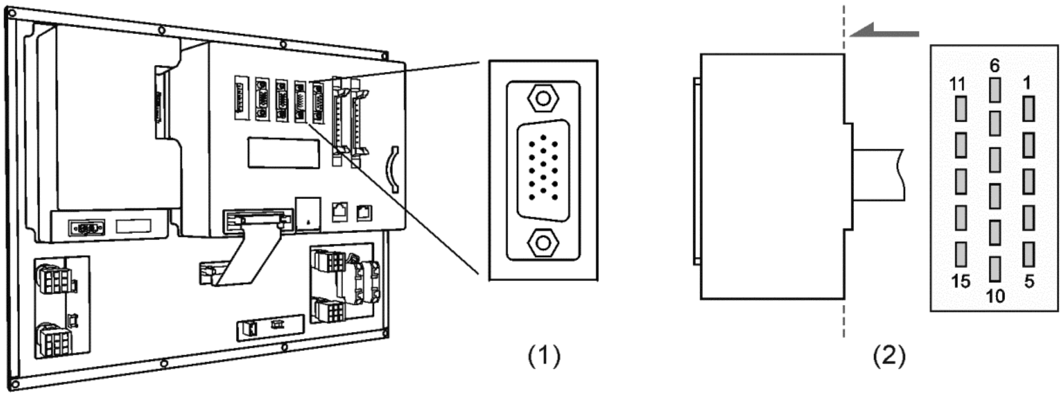
(1) RS-485 connector on NC controller (Female); (2) RS-485 connector (Male)

Model	Terminal ID	Pin No	Function
NC_ _EM	SPINDLE	PIN 2	D+
		PIN 3	D-

3.5 Wiring for AXIS 1 ~ 4 connector

3

The NC series controller features positive and negative hardware limits and origin sensor inputs for 4 axes at the AXIS 1~4 connector with 12 input points in total. Users may set the input signal to 0 V or +24 VDC depending upon whether COM is connected to +24 VDC or 0 V.
See below for NC3_ _ series PIN assignment.



(1) AXIS1 ~ 4 connector on NC controller (Female); (2) AXIS1 ~ 4 connector (Male)

Model	Terminal ID	Pin No	Function	
		PIN 1	OT0+	Positive limit of Axis 1
		PIN 2	OT0-	Negative limit of Axis 1
		PIN 3	DOG0	Origin of Axis 1
		PIN 4	OT0+	Positive limit of Axis 2
		PIN 5	OT0-	Negative limit of Axis 2
		PIN 6	DOG0	Origin of Axis 2
		PIN 7	OT0+	Positive limit of Axis 3
NC3_ _	AXIS 1~4	PIN 8	OT0-	Negative limit of Axis 3
		PIN 9	DOG0	Origin of Axis 3
		PIN 10	OT0+	Positive limit of Axis 4
		PIN 11	OT0-	Negative limit of Axis 4
		PIN 12	DOG0	Origin of Axis 4
		PIN 13	COM	COM
		PIN 14	COM	COM
		PIN 15	COM	COM

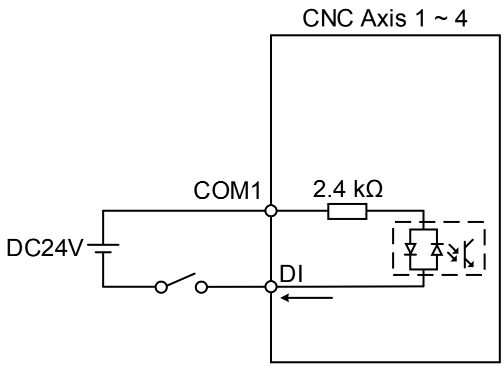
For 6-axis servo applications, set parameter 49 to 1. In this case, PIN10 will be the home point (DOG3) of Axis 4; PIN11 will be the home point (DOG4) of Axis 5; and PIN12 will be the home point (DOG5) of Axis 6.

Description of Special M:

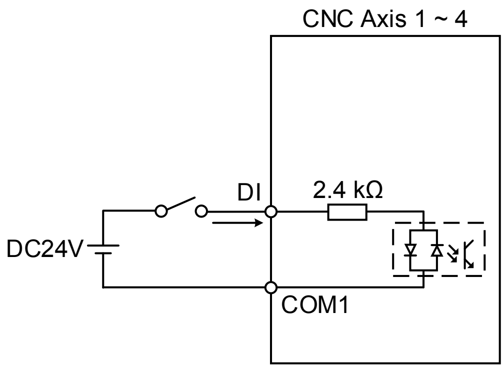
P1 ~ P3	Digital inputs for positive limit, negative limit, and home point of Axis 1 (M2144, M2145, M2146)
P4 ~ P6	Digital inputs for positive limit, negative limit, and home point of Axis 2 (M2148, M2149, M2150)
P7 ~ P9	Digital inputs for positive limit, negative limit, and home point of Axis 3 (M2152, M2153, M2154)
P10 ~ P12	Digital inputs for positive limit, negative limit, and home point of Axis 4 (M2156, M2157, M2158)

DI wiring with external power supply (Allowable voltage: 17 ~ 32 VDC; surge current: less than 50 mA). See the wiring below:

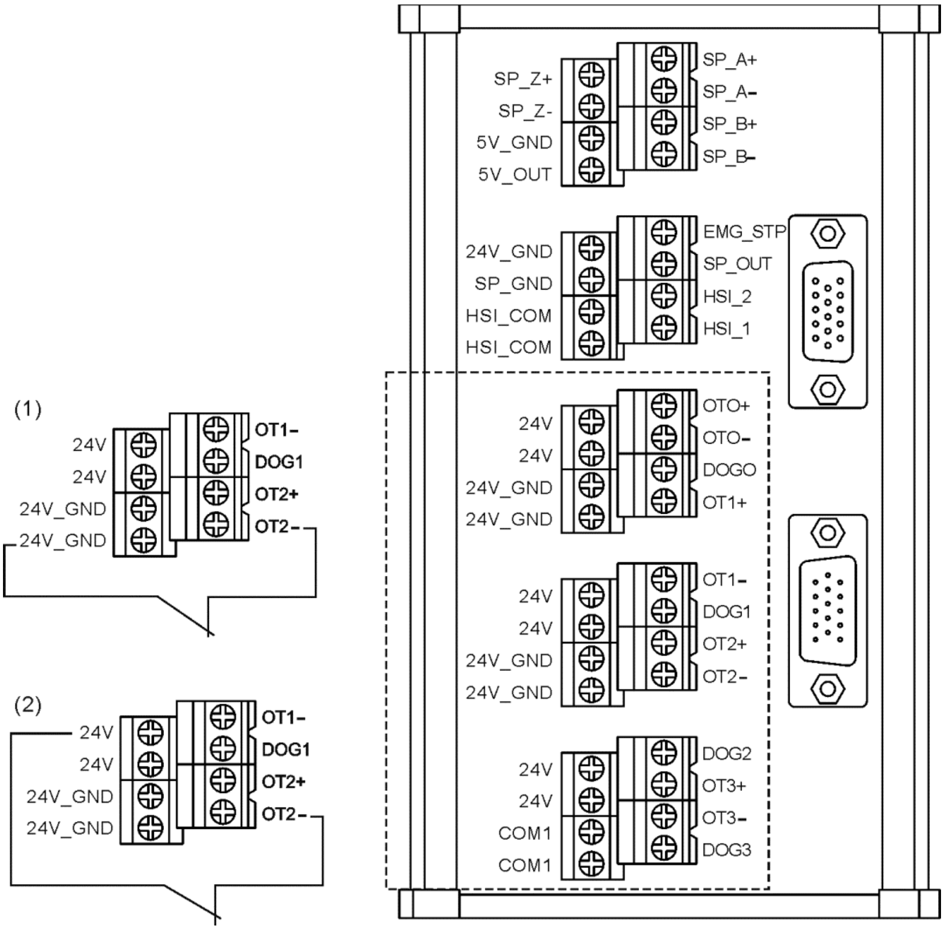
NPN transistor (SINK mode)



PNP transistor (SOURCE mode)



Wiring for Conversion Card (NC-EXM-S01) which connects to AXIS 1 ~ 4 Connector



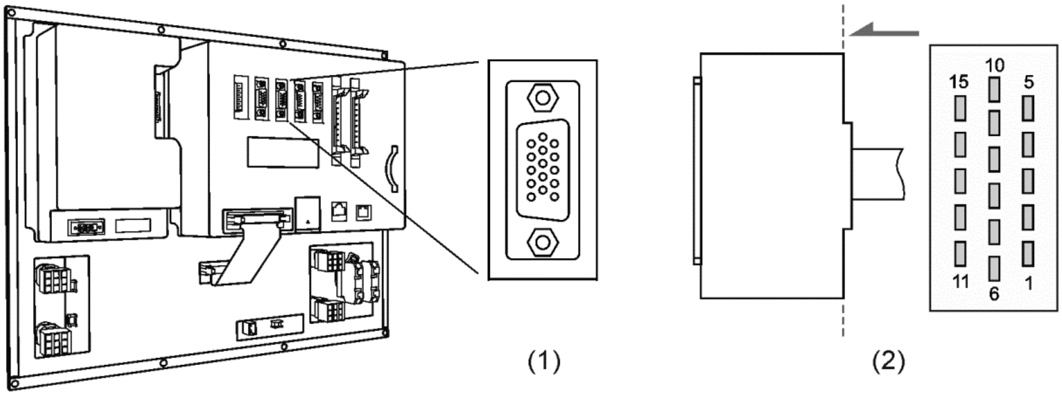
(1) DI input when COM1 connects to 0 V; (2) DI input when COM1 connects to +24 VDC.

3

3.6 Wiring for spindle

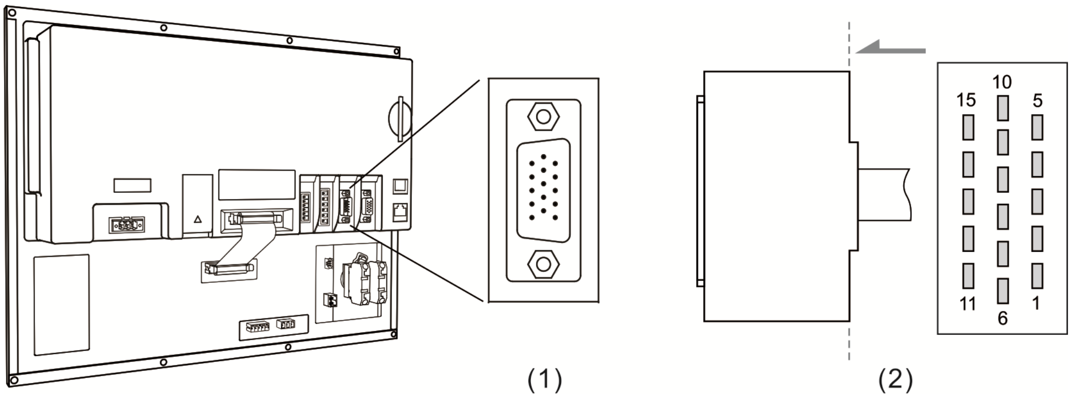
NC series controller features one set of spindle feedback input.

See below for NC3_ _ series PIN assignment.



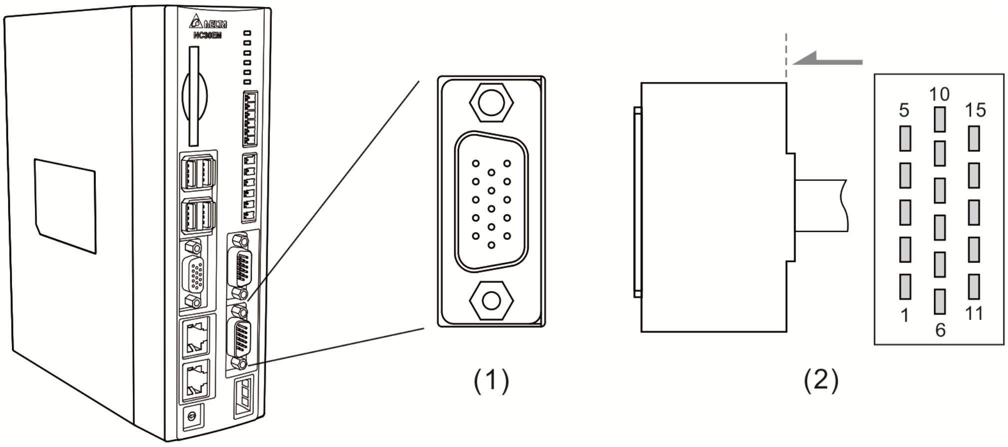
(1) SPINDLE connector on NC controller (Female); (2) SPINDLE connector (Male)

See below for NC2_ _ series PIN assignment.



(1) SPINDLE connector on NC controller (Female); (2) SPINDLE connector (Male)

See below for NC_ _ EM series PIN assignment.



(1) SPINDLE connector on NC controller (Female); (2) SPINDLE connector (Male)

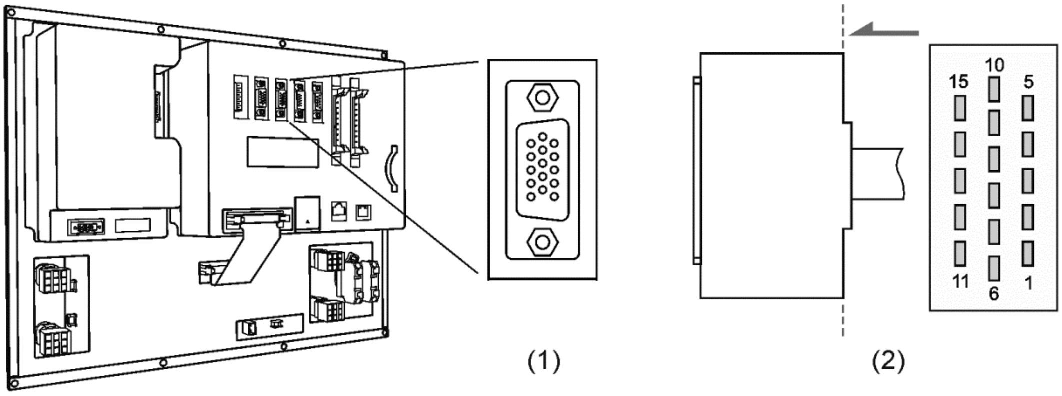
See the description below.

Model	Terminal ID	Pin No	Function	
ALL	SPINDLE	PIN 8	SP_A+	Spindle encoder A+ phase input
		PIN 9	SP_A-	Spindle encoder A- phase input
		PIN 10	DC +5V_OUT	Spindle encoder power output (+5 V_{DC})
		PIN 11	SP_B+	Spindle encoder B+ phase input
		PIN 12	SP_B-	Spindle encoder B- phase input
		PIN 13	SP_Z+	Spindle encoder Z+ phase input
		PIN 14	SP_Z-	Spindle encoder Z- phase input
		PIN 15	GND	Spindle encoder power output (0 V)

3.7 Wiring for analog spindle connector

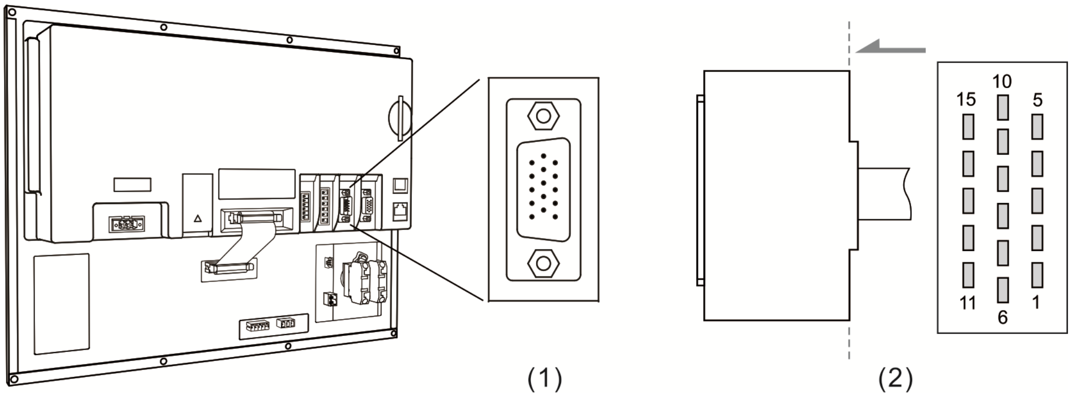
NC series controller features one spindle analog output for controlling spindle speed.

See below for NC3_ _ series PIN assignment.



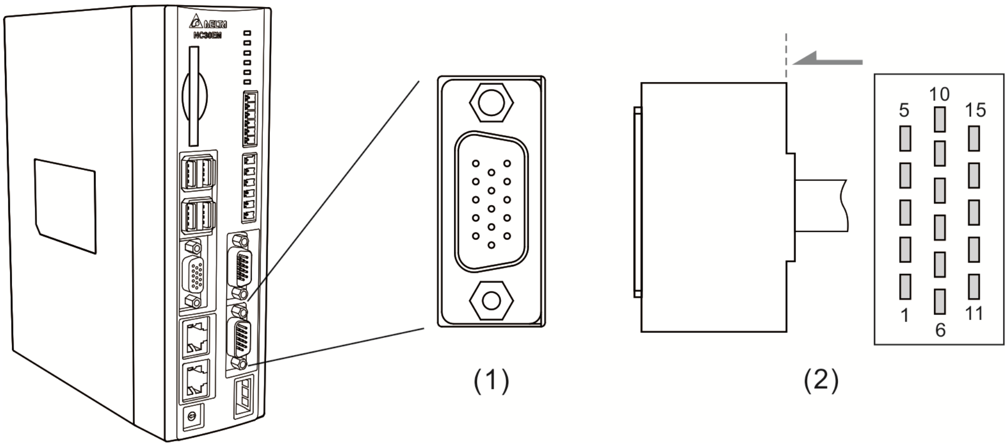
(1) Spindle connector on NC controller (Female); (2) Spindle connector (Male)

See below for NC2_ _ series PIN assignment.



(1) Spindle connector on NC controller (Female); (2) Spindle connector (Male)

See below for NC_ _ EM series PIN assignment.



(1) Spindle connector on NC controller (Female); (2) Spindle connector (Male)

See the description below:

Model	Terminal ID	Pin No	Function	
ALL	SPINDLE	PIN 4	SP_OUT	Spindle analog output
		PIN 5	SP_GND	Spindle analog ground

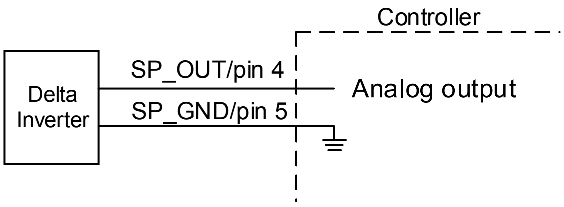
Settings for analog spindle

Step 1: Setting for Parameter 399. Servo spindle: set to 1101; Analog spindle: set to 1020

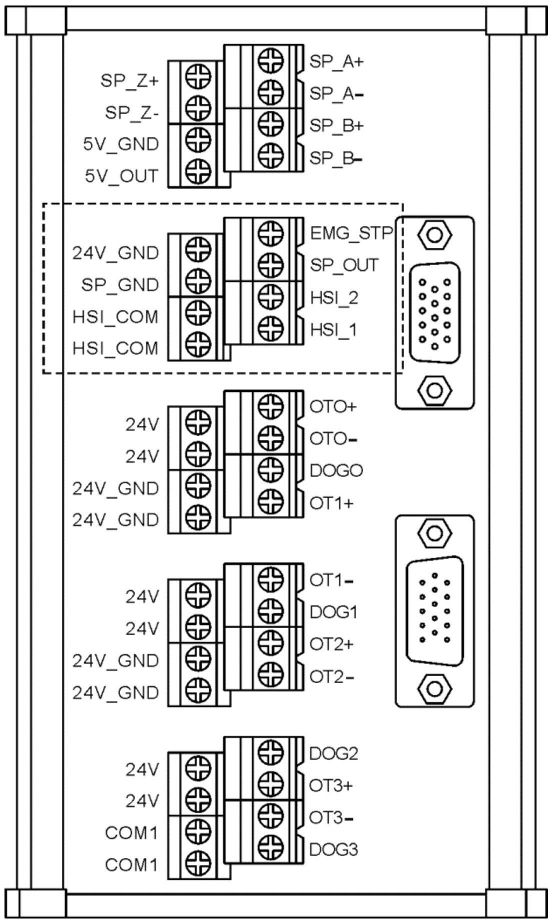
Bit	Description	Setting range
0	Spindle function 0: : spindle OFF / 1: spindle ON	0 ~ 1
1	Close loop control flag 0: close loop control OFF / 1: Close loop control ON (feedback encoder is required)	0 ~ 1
2 ~ 3	Spindle output mode 0: DMCNET (servo spindle) / 2: EDAC (analog output)	0 ~ 2
4	Speed control mode 0: rpm / 1: PUU	0 ~ 1
5	Feedback form 0: high resolution (x1000) / 1: normal resolution (x4)	0 ~ 1

Step 2: The SP1 in the channel setting has to be enabled and set to 10.

Step 3: When using analog spindle output, the wiring pin will be Pin 4 and Pin 5. The 0 ~ 10 VDC of analog output corresponds to S0 ~ S (maximum speed). (The resolution of -10 V ~ +10 V is 14-bit.)



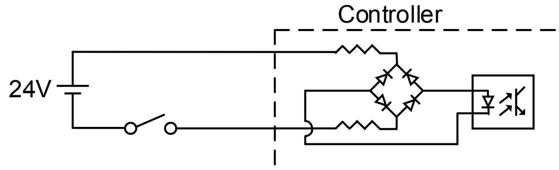
Conversion Card (NC-EXM-S01)

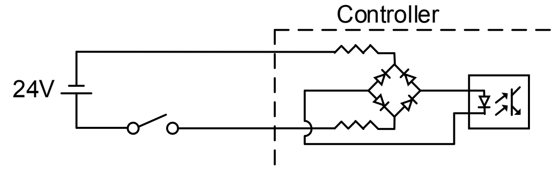


3.8 Wiring for HSI high speed counter connector

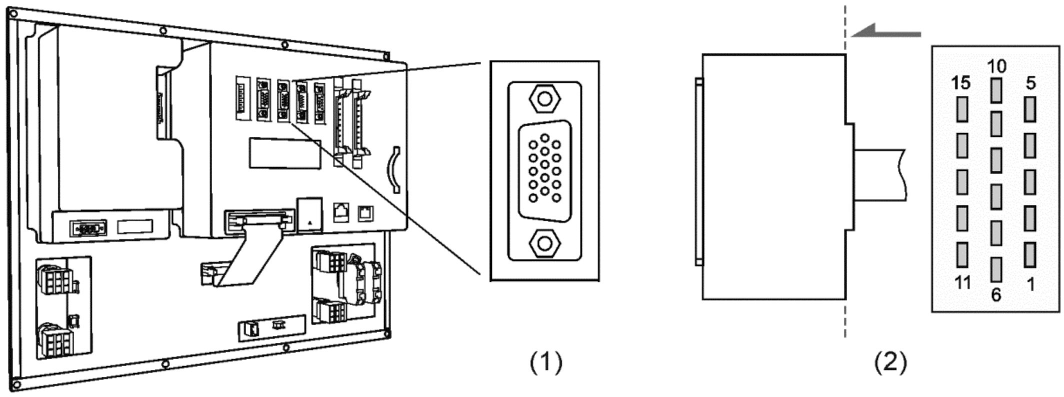
NC series controller features 2 high-speed counter inputs. Below is the DI wiring for high-speed counter, of which max. input bandwidth is up to 5 MHz. It can also be used for G31 interruption input with external power supply (voltage: 22 ~ 26 V; permissible current: 8 ~ 20 mA; surge current: below 50 mA).

Setting HIS_1 for G31 interruption input: parameter 46Bit5 = 1; parameter 307Bit4 = 1; input limit parameter 25Bit0 = 1; special M = M2142.





See below for NC3_ _ series PIN assignment.

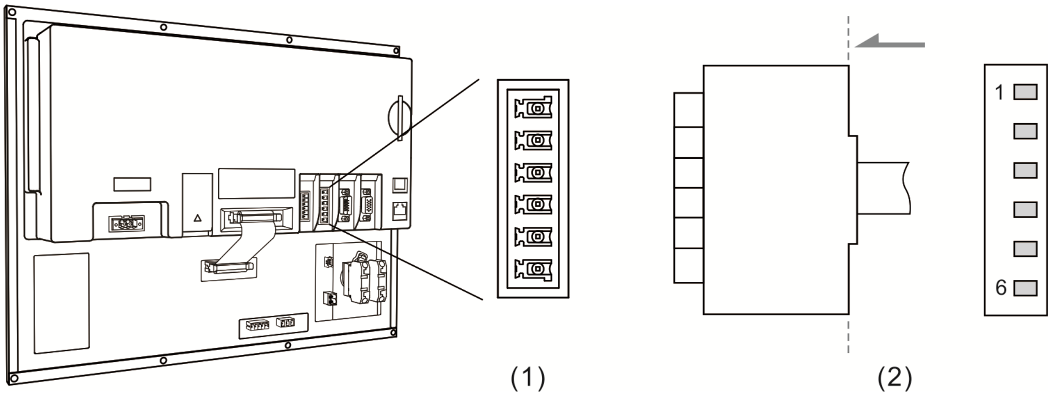


(1) HSI connector on NC controller (Female); (2) HSI connector (Male)

See the description below.

Model	Terminal ID	Pin No	Function	
NC3_ _	SPINDLE	PIN 1	HSI_COM	High-speed counter COM, for +24 VDC or 0 V
		PIN 2	HSI_1	High-speed counter input 1 (10 mA)
		PIN 3	HSI_2	High-speed counter input 2 (10 mA)

See below for NC2_ _ series PIN assignment.

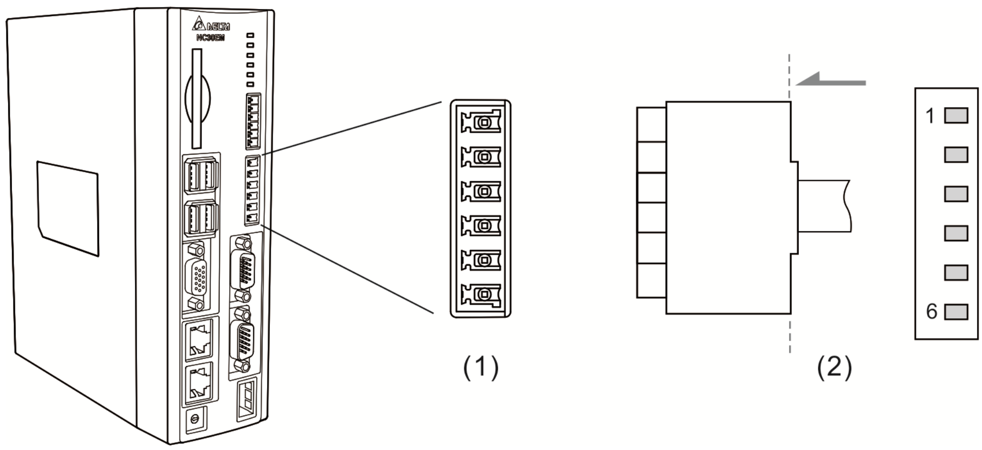


(1) HSI connector on NC controller (Female); (2) HSI connector (Male)

See the description below.

Model	Terminal ID	Pin No	Function	
NC2_ _	HSI	PIN 3	HSI_1	High-speed counter input 1 (10 mA)
		PIN 4	HSI_COM	High-speed counter COM, for +24 VDC or 0 V
		PIN 5	HSI_2	High-speed counter input 2 (10 mA)
		PIN 6	HSI_COM	High-speed counter COM, short-circuits with PIN 4

See below for NC_ _ EM series PIN assignment.



(1) HSI connector on NC controller (Female); (2) HSI connector (Male)

See the description below.

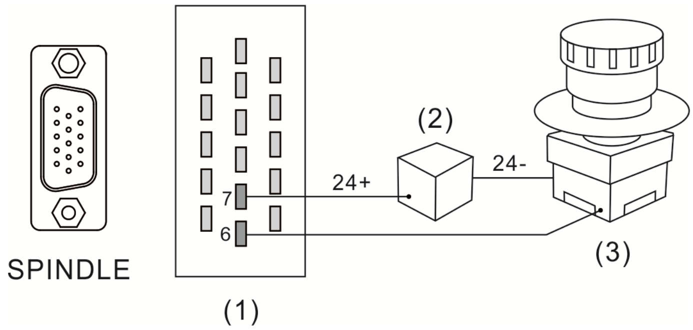
Model	Terminal ID	Pin No	Function	
NC__EM	HSI	PIN 3	HSI_1	High-speed counter input 1 (10 mA)
		PIN 4	GND	GND
		PIN 5	HSI_2	High-speed counter input 2 (10 mA)
		PIN 6	HSI_COM	High-speed counter COM, for +24 VDC or 0 V

Note: HSI_1 and HSI_2 have +5V output, both can directly connect to GND.

3.9 Wiring for EMG (Emergency Stop) connector

NC series controller features one set of EMG signal input connector.

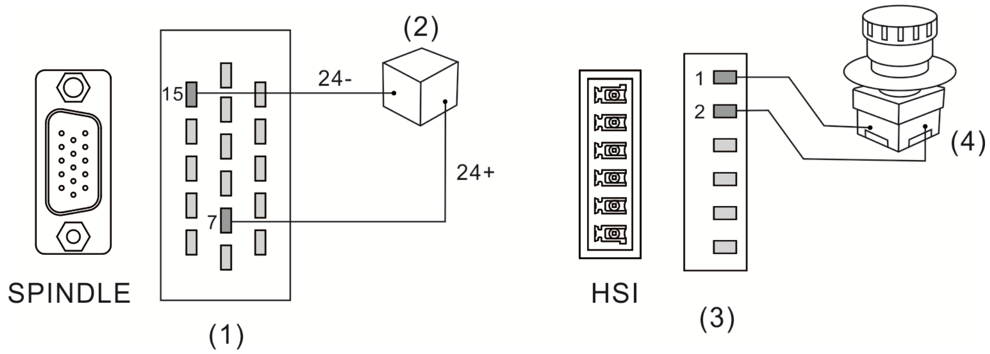
See below for NC3_ _ series PIN assignment.



(1) EMG connector (Female); (2) Power supply; (3) Emergency stop button

Model	Terminal ID	Pin No	Function	
NC3_ _	SPINDLE	PIN 6	EMG_GND	Emergency stop input
		PIN 7	EMG_IN	+24 VDC input

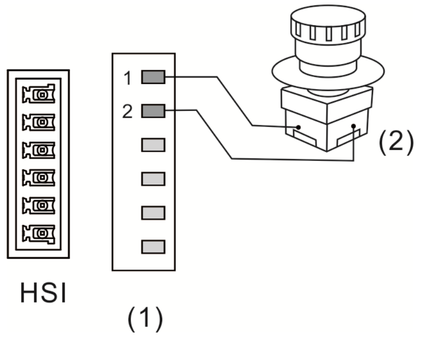
See below for NC2_ _ series PIN assignment.



(1) & (3) EMG connector (Female); (2) Power supply; (4) Emergency stop button

Model	Terminal ID	Pin No	Function	
NC2_ _	SPINDLE	PIN 7	EMG_IN	Power input for emergency stop, +24 VDC
		PIN 15	GND	Power input for emergency stop, 0 VDC
	HSI	PIN 1	EMG_IN	Emergency stop input
		PIN 2	EMG_GND	Emergency stop input

See below for NC_ _ EM series PIN assignment.



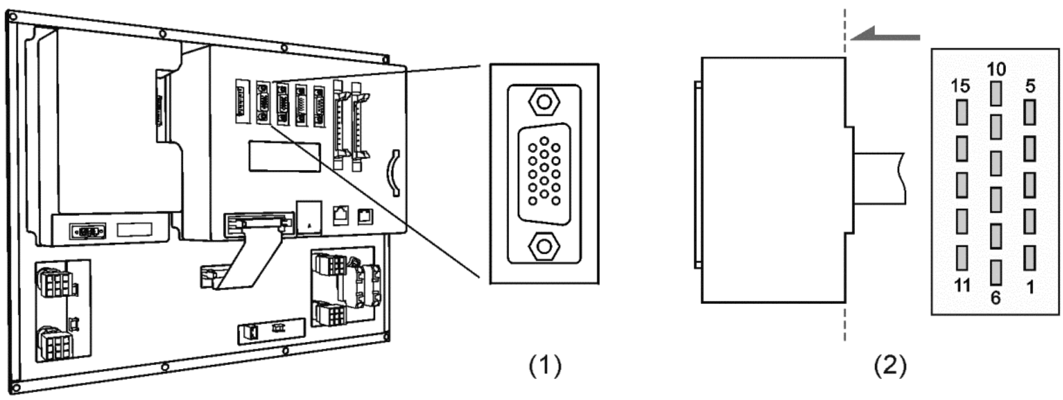
(1) EMG connector (Female); (2) Emergency stop button

Model	Terminal ID	Pin No	Function	
NC_ _EM	HSI	PIN 1	EMG_IN	EMG (+5 VDC output)
		PIN 6	HSI_COM	GND

3.10 Wiring for MPG

The NC series controller features one MPG connector for MPG applications. This connector supplies +5 VDC working power and can power the MPG device directly.

See below for NC3_ _ series PIN assignment.



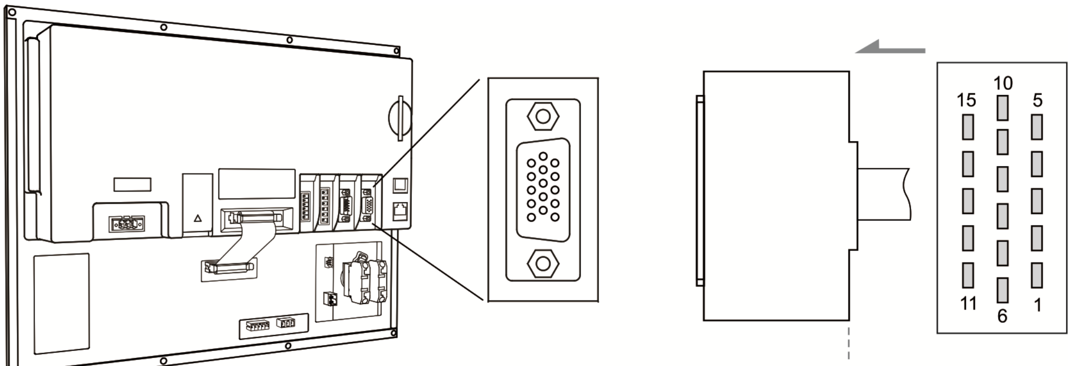
(1) MPG connector on NC controller (Female); (2) MPG connector (Male)

See the description below:

Model	Terminal ID	Pin No	Function	
NC3_ _	MPG	PIN 1	DI_COM	DI_COM, which can be connected to +24 VDC or 0 V
		PIN 2	DI_1	(X28) X-axis
		PIN 3	DI_2	(X29) Y-axis
		PIN 4	DI_3	(X30) Z-axis

Model	Terminal ID	Pin No	Function	
		PIN 5	DI_4	(X31) magnification x1
		PIN 6	DI_5	(X32) magnification x10
		PIN 7	DI_6	(X33) magnification x100
		PIN 8	GND	GND, short-circuits with PIN 9 and PIN 15
		PIN 9	GND	GND, short-circuits with PIN 8 and PIN 15
		PIN 10	DC +5V_OUT	DC +5V_OUT
		PIN 11	XA+	XA+
		PIN 12	XA-	XA-
		PIN 13	XB+	XB+
		PIN 14	XB-	XB-
		PIN 15	GND	GND, short-circuits with PIN 8 and PIN 9

See below for NC2_ _ series PIN assignment.



(1) (2)

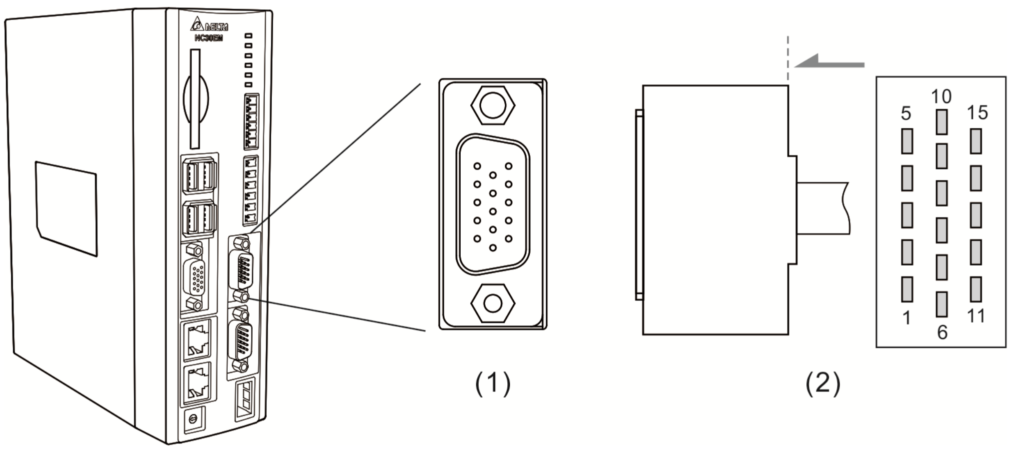
(1) MPG connector on NC controller (Female); (2) MPG connector (Male)

See the description below:

Model	Terminal ID	Pin No	Function	
NC2_ _	MPG	PIN 1	DI_COM	DI_COM, which can be connected to +24 VDC or 0 V
		PIN 2	DI_1	(X28) X-axis
		PIN 3	DI_2	(X29) Y-axis
		PIN 4	DI_3	(X30) Z-axis
		PIN 5	DI_4	(X31) magnification x1
		PIN 6	DI_5	(X32) magnification x10
		PIN 7	DI_6	(X33) magnification x100

Model	Terminal ID	Pin No	Function	
		PIN 8	DI_7	(X26) Z-axis
		PIN 9	DO_8	Y27
		PIN 10	DC +5V_OUT	DC +5V_OUT
		PIN 11	XA+	XA+
		PIN 12	XA-	XA-
		PIN 13	XB+	XB+
		PIN 14	XB-	XB-
		PIN 15	GND	GND

See below for NC_ _ EM series PIN assignment.



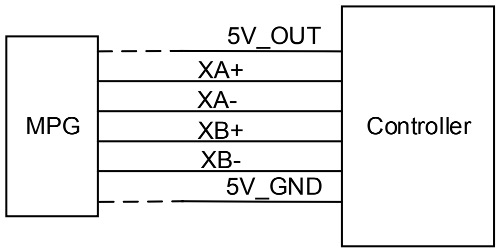
(1) MPG connector on NC controller (Female); (2) MPG connector (Male)

See the description below:

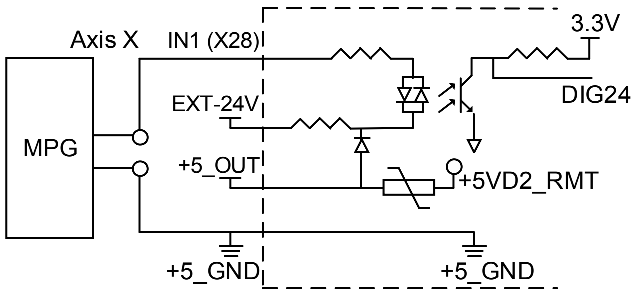
Model	Terminal ID	Pin No	Function	
NC_ _EM	MPG	PIN 1	DI_COM	DI_COM, which can be connected to +24 VDC or 0 V
		PIN 2	DI_1	X0
		PIN 3	DI_2	X1
		PIN 4	DI_3	X2
		PIN 5	DI_4	X3
		PIN 6	DI_5	X4
		PIN 7	DI_6	X5
		PIN 8	DI_7	X6
		PIN 9	DI_8	X7
		PIN 10	DC +5V_OUT	DC +5V_OUT

Model	Terminal ID	Pin No	Function	
		PIN 11	XA+	XA+
		PIN 12	XA-	XA-
		PIN 13	XB+	XB+
		PIN 14	XB-	XB-
		PIN 15	GND	GND

MPG pulse input wiring with internal power of 5 VDC

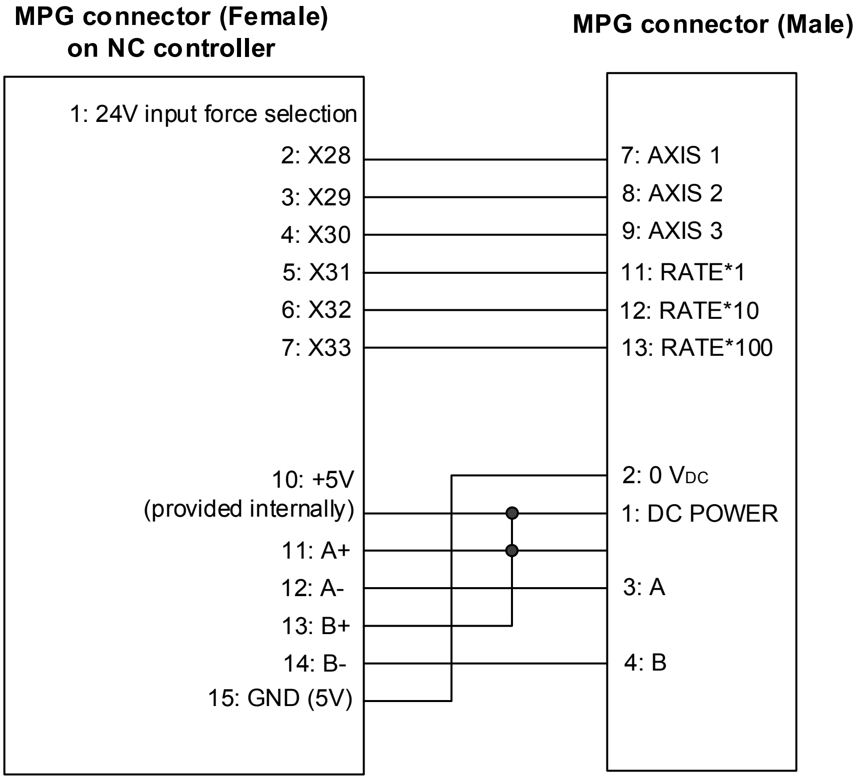


DI pin wiring:

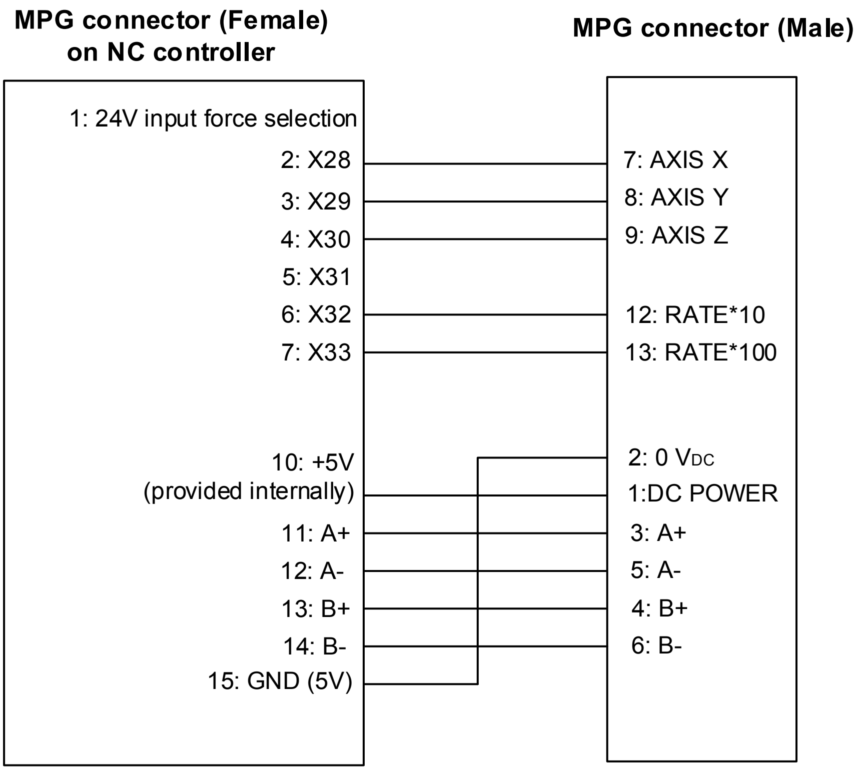


The wiring varies with the signal types of the MPG (only support 100 PPR type), which includes single-ended (EHDW-BA6SI) and differential (EHDWBE6SI).

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

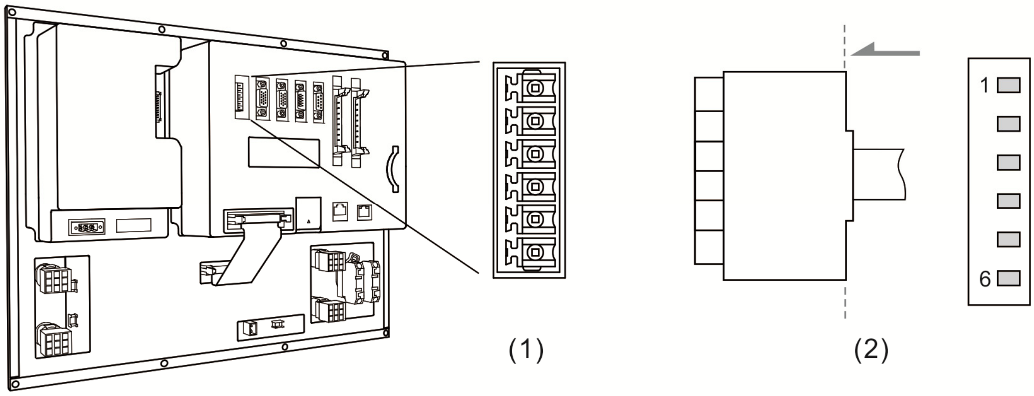


Wiring for differential type MPG (EHDW-BE6SI):



3.11 Wiring for Remote I/O

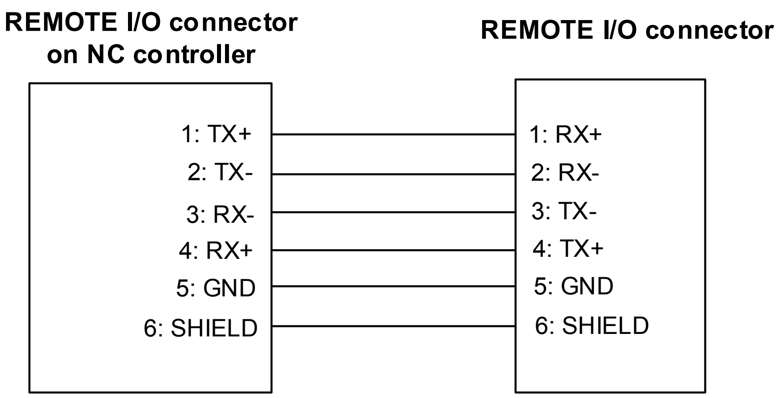
NC series controller provides REMOTE I/O connector which can be serial connected to up to 8 expansion modules with 256 input and 256 output points. See its pin assignment and illustration below.



(1) Remote I/O connector on NC controller (Female); (2) Remote I/O connector on the module (Male)

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

Remote I/O module:



REMOTE I/O module has two types: NC-EIO-T3232 (Optical coupler) and NC-EIO-R3216 (Relay)

1. NC-EIO-T3232 (Optical coupler)

The optical coupler type remote I/O module is connected to NC300 as the remote I/O which applies RS-422 communication protocol. Stations can be selected on the board. The first station starts from X256/Y256. The second station will be from X288/Y288 and so on. Address of every additional station will offset by 32 points. Users can cascade up to 8 modules with total 256

3

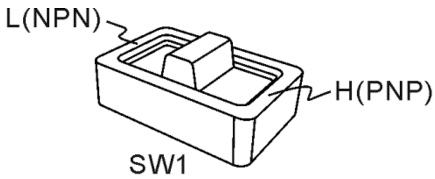
points for each I/O points.

2. NC-EIO-R3216 (Relay)

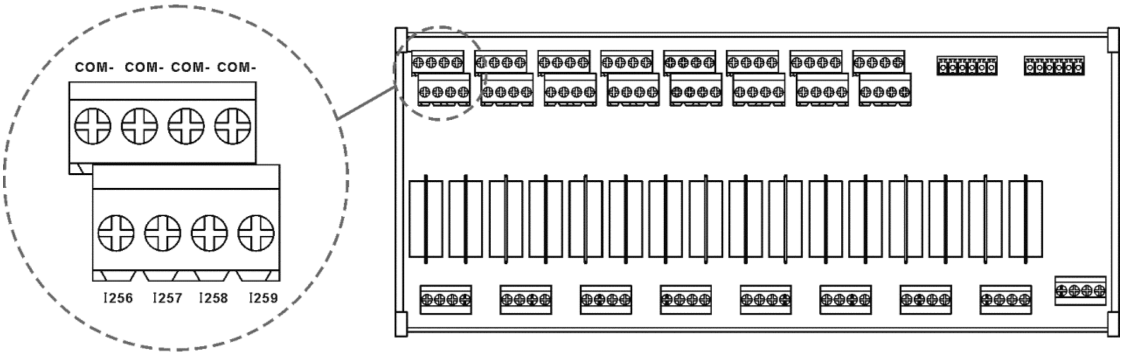
The relay type remote I/O module is connected to NC300 as the remote I/O which applies RS-422 communication protocol. Stations can be selected on the board. The first station starts from X256/Y256. The second station will be X288/Y288 and so on. Address of every additional station will offset by 32 points. Address of every additional station will offset by 32 points. This module supports 32 DI points and 16 DO points. The rest 16 points address will not be used and the next station still offsets by 32 points for Y address.

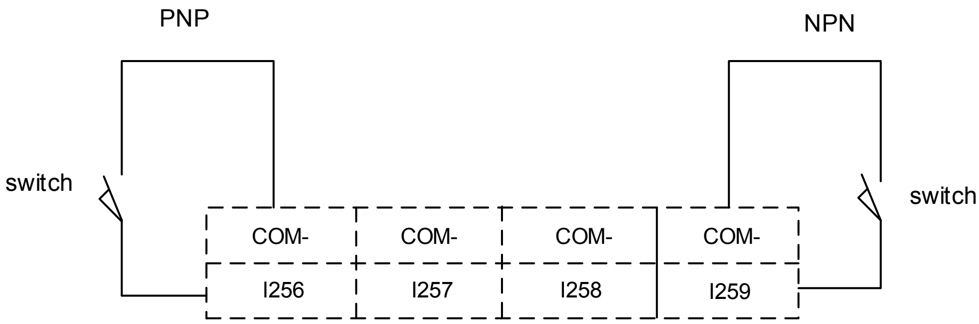
Wiring of remote I/O module

COM- is for signal current and it is prohibited to connect to 24 VDC or 0 V power. PNP and NPN type of external input can be selected by switching to H and L.



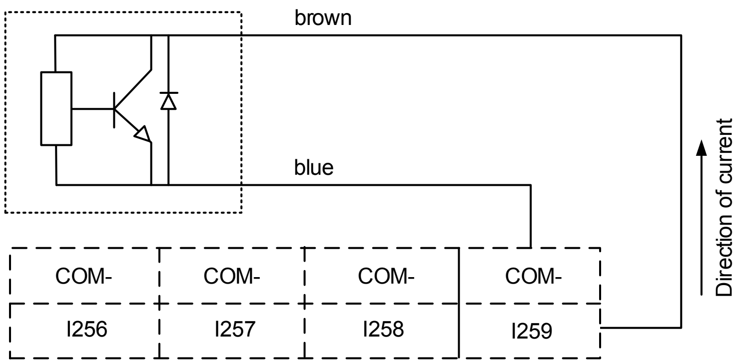
Wiring example: The button and mechanical switch





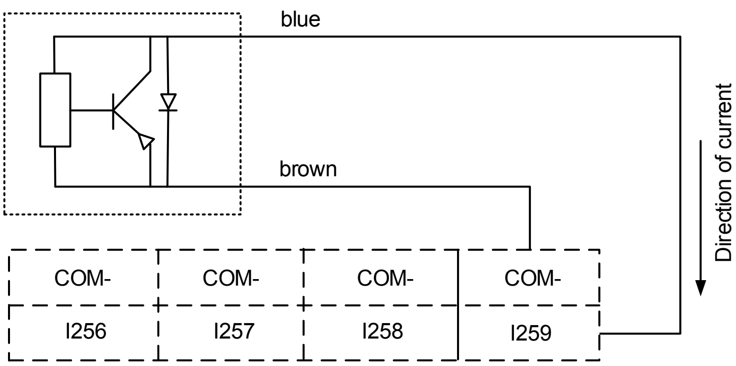
Wiring example: The proximity switch of NPN two-wire system

NPN 2 wire system proximity switch



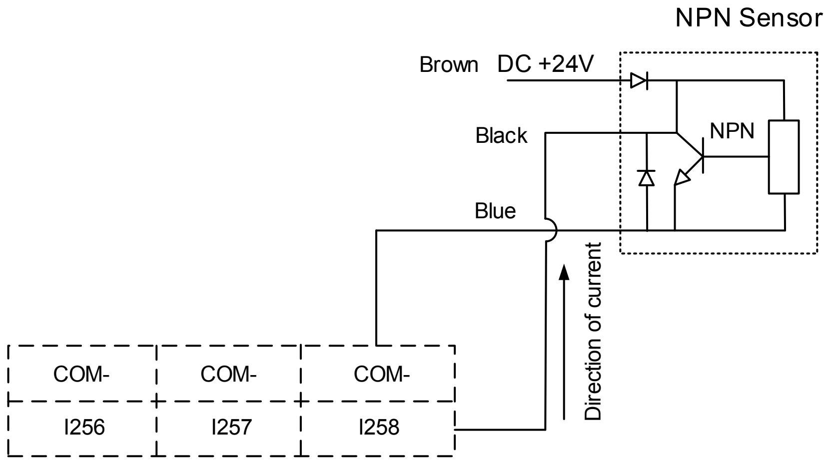
Wiring example: The proximity switch of PNP two-wire system

PNP 2 wire system proximity switch

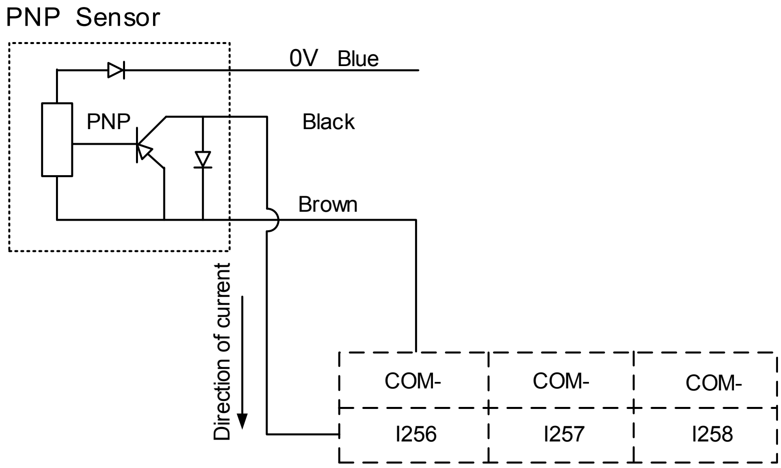


3

Wring example: The proximity switch of NPN three-wire system



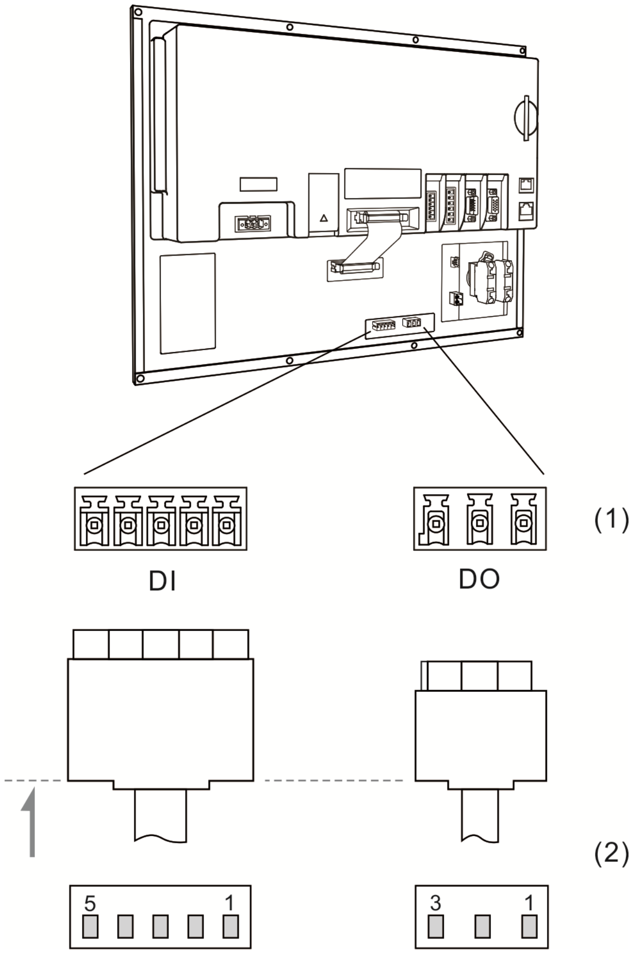
Wring example: The proximity switch of PNP three-wire system



3.12 Wiring for local I/O connector

For providing a more flexible I/O configuration, NC series controller provides external I/O port.

NC200A-MI-A



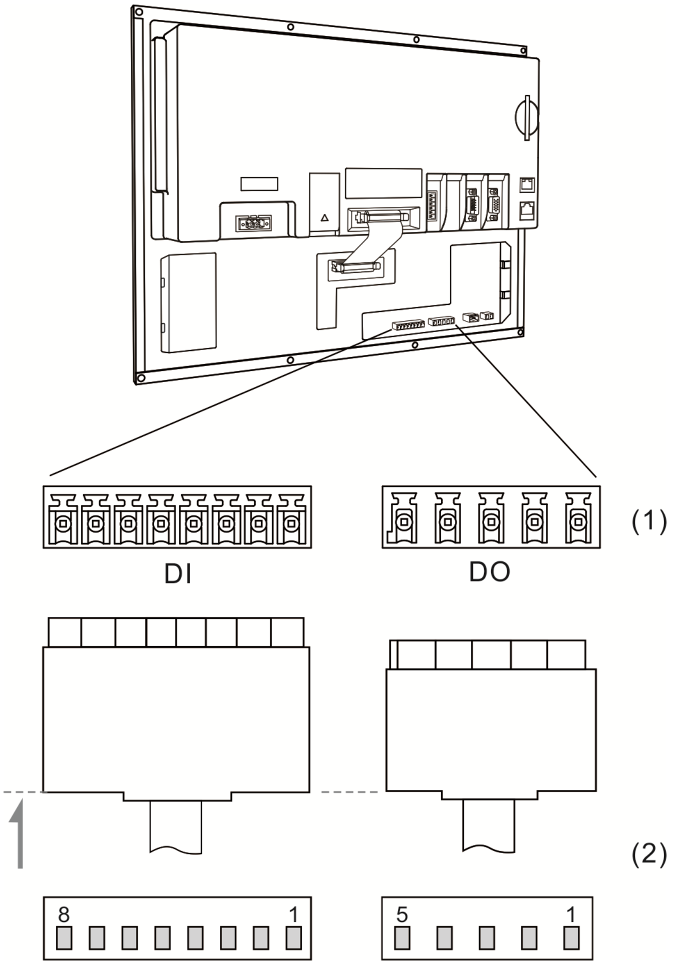
(1) Local I/O connector on NC controller (Female); (2) Local I/O connector (Male)

DI			
PIN 1	X112	PIN 2	X113
PIN 3	X114	PIN 4	X115
PIN 5	X116		

DO			
PIN 1	Y112	PIN 2	Y113
PIN 3	Y114		

3

NC200A-LI-A, NC200P-LI-A

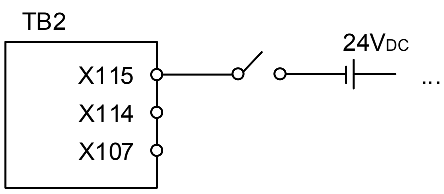


(1) Local I/O connector on NC controller (Female); (2) Local I/O connector (Male)

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

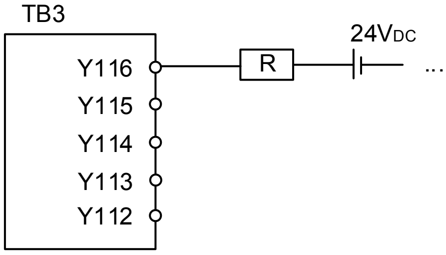
DO			
PIN 1	Y112	PIN 2	Y113
PIN 3	Y114	PIN 4	Y115
PIN 5	Y116		

DI wiring, external power supply



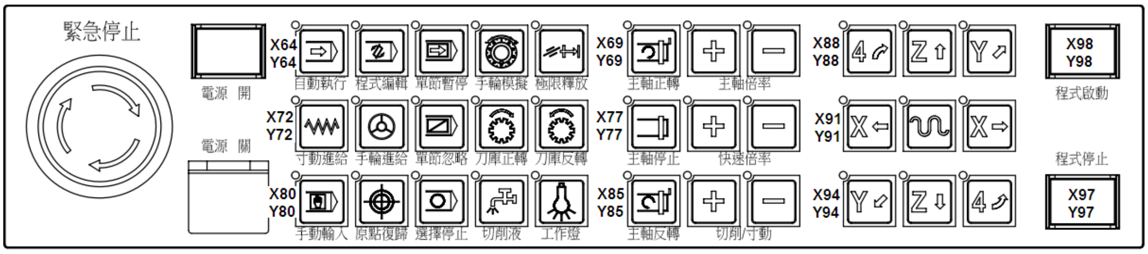
3

DO wiring, external power supply

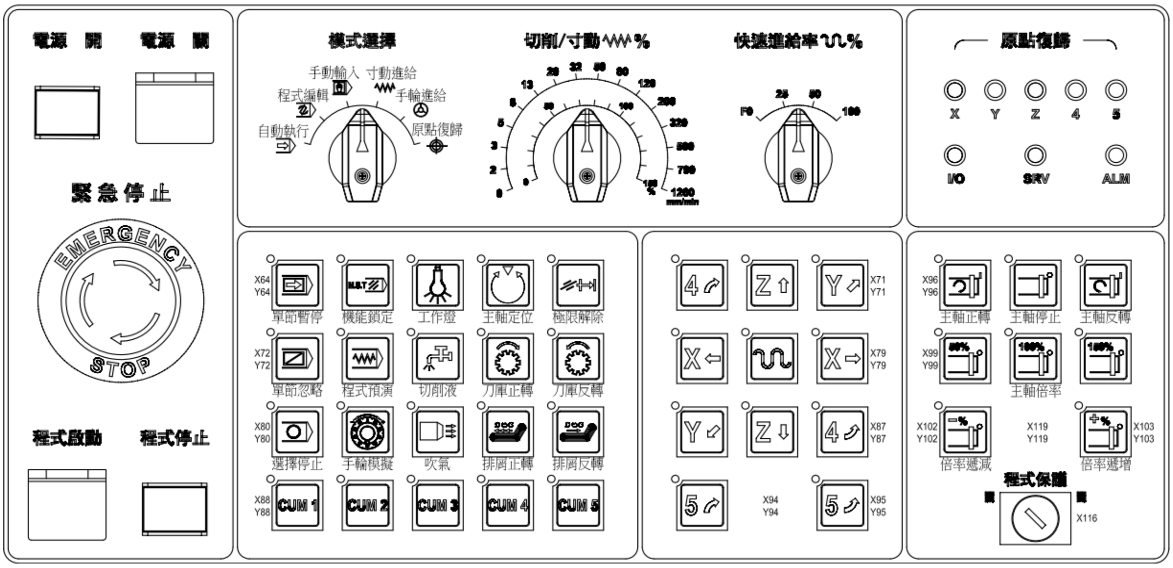


3.13 I/O assignment of secondary control panel - NC300 / NC310 series

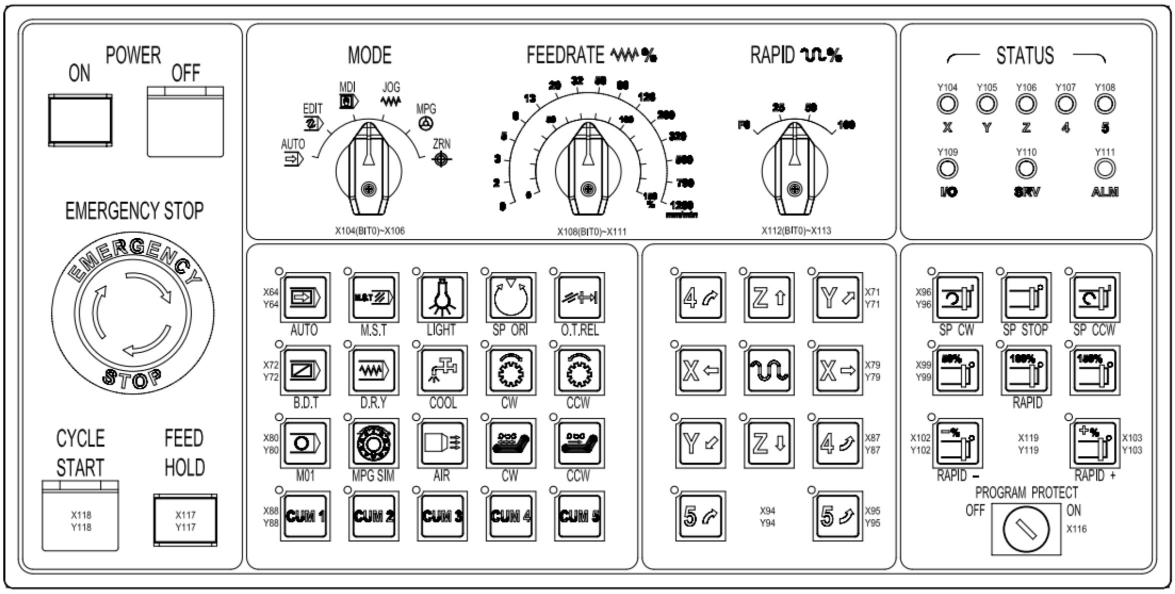
NC300A-MI-A/AE (all-in-one model)



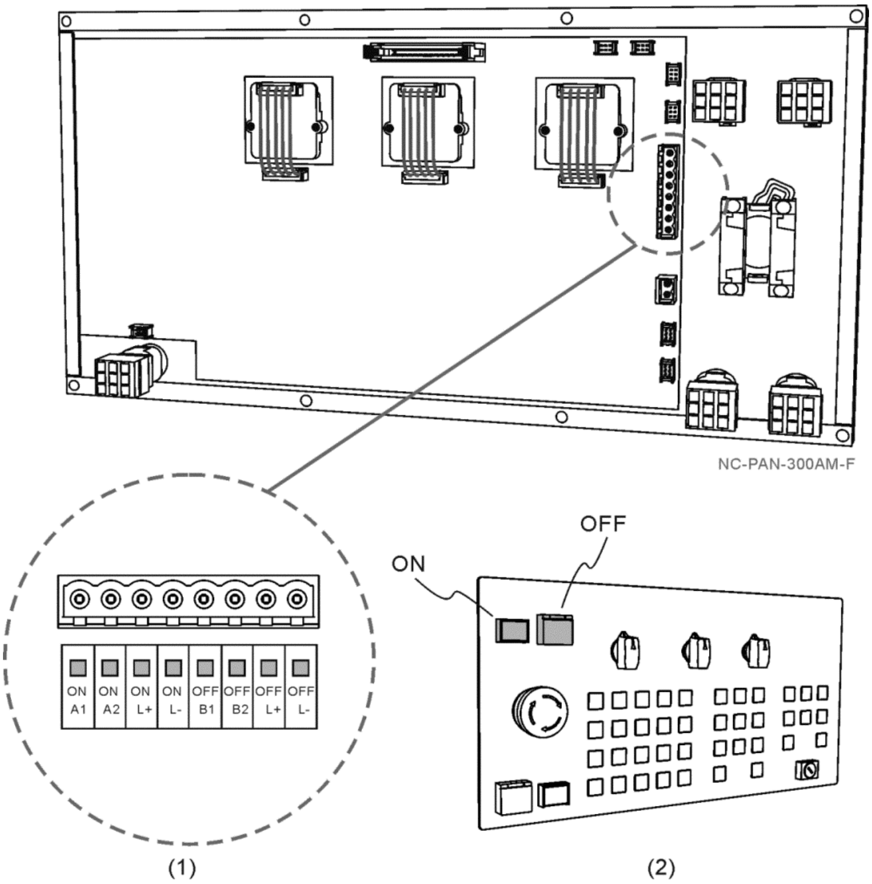
NC-PAN-300AM-F (P)



NC-PAN-300AM-F(P)E

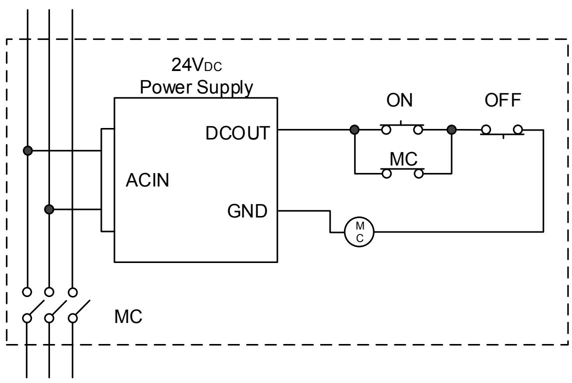


Wiring for terminal block of Power ON/OFF



(1) Terminal block description:

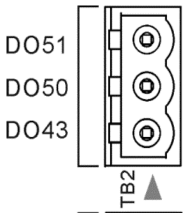
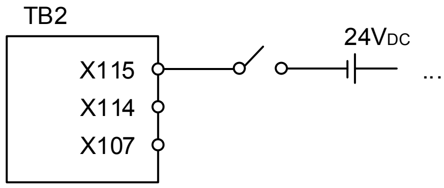
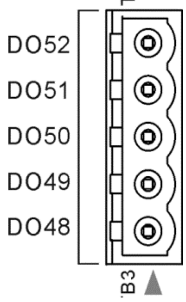
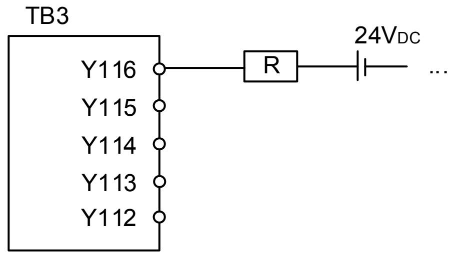
To switch on the Power ON Indicator, supply +24 VDC power to PIN ONL+ and 0V to ONL-; to switch on the Power OFF indicator, supply +24 VDC power to OFFL+ and 0V to OFFL-.



If the Power ON button is pressed, the circuit is closed between PIN ONA1 and ONA2; if the Power OFF button is pressed, the circuit is open between PIN OFFB1 and OFFB2.

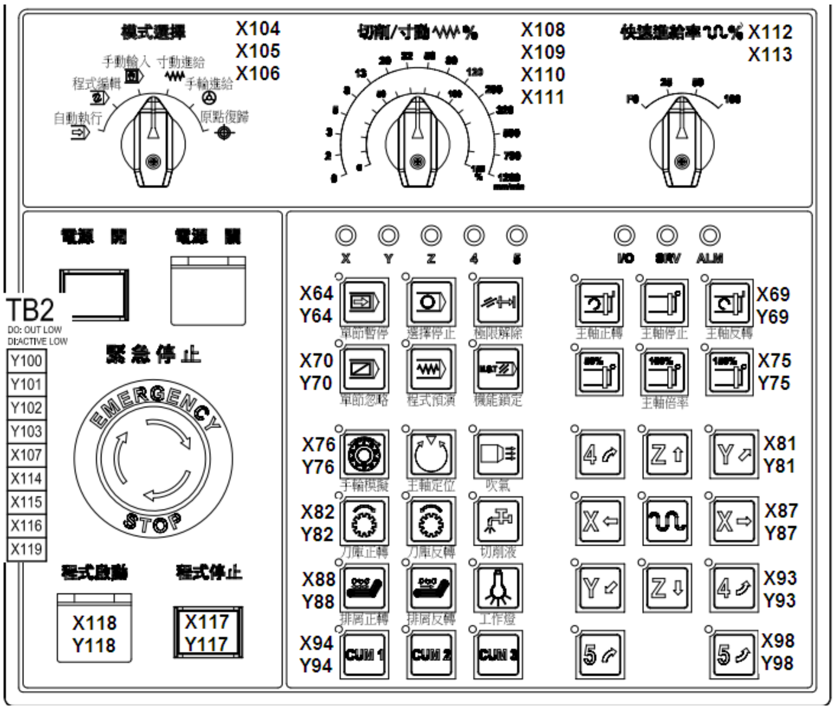
(2) Front view of Power ON/OFF buttons

I/O in NC-PAN-300AM-P

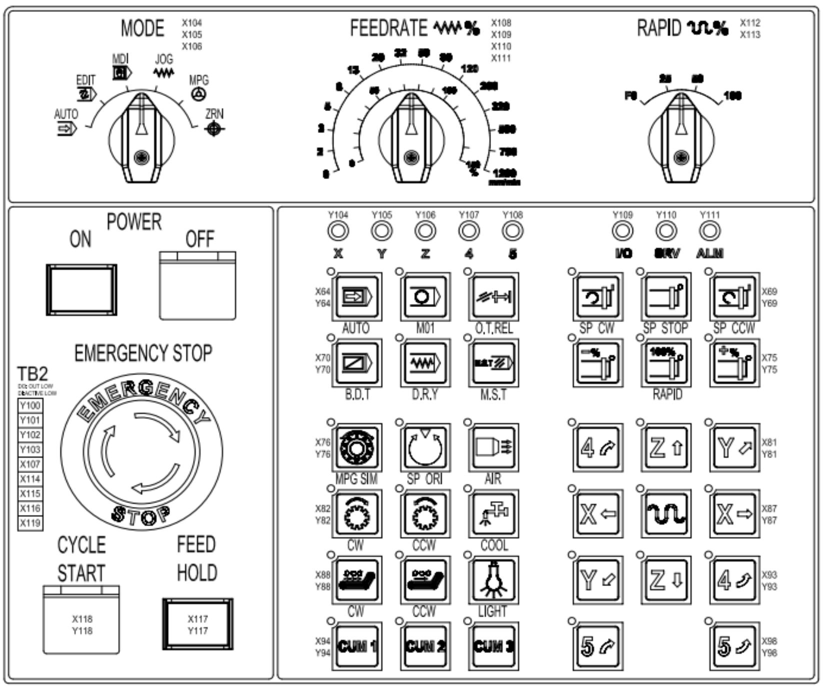
	DI signal (allowable current: 8 ~ 25 mA; surge current: under 40 mA) 
	DO driven load (allowable current: under 40 mA; surge current: under 100 mA) 

3.14 I/O assignment of secondary control panel - NC311 series

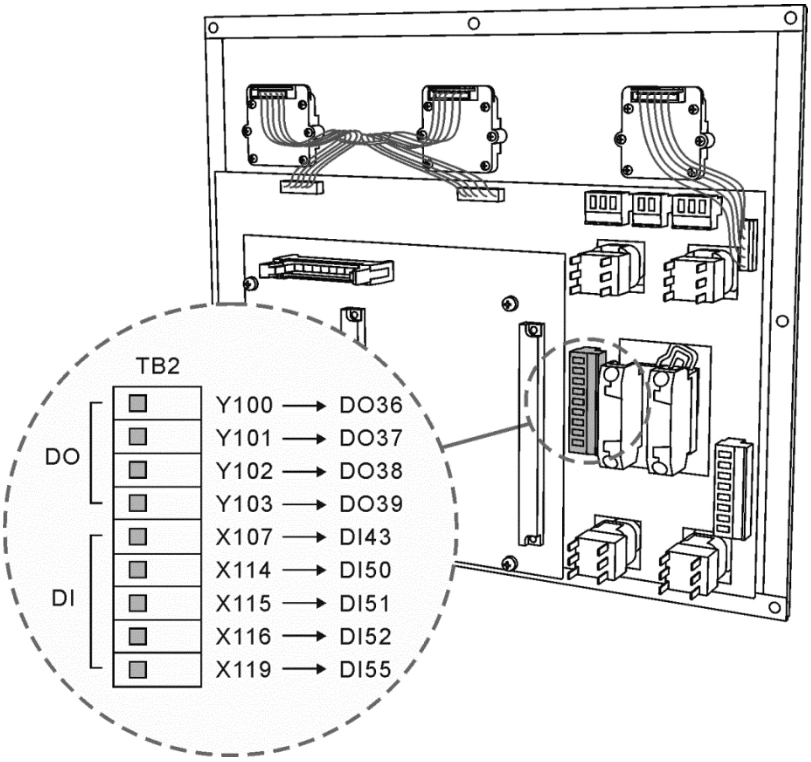
NC-PAN-311AM-F (P)



NC-PAN-311AM-F (P) E



Wiring for terminal block



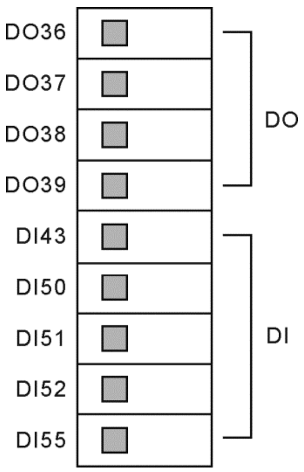
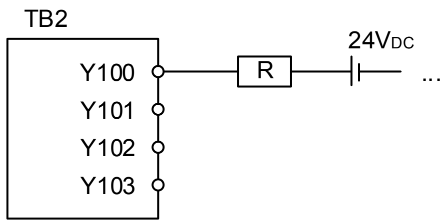
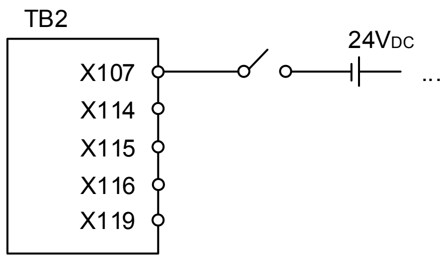
<table>
<tr>
<td rowspan="2">
</td>
<td>DO general load (allowable current: under 40 mA; surge current: under 100 mA)

</td>
</tr>
<tr>
<td>DI signal (allowable current: 8 ~ 25 mA; surge current: under 40 mA)

</td>
</tr>
</table>

Table of Group Menu

4

This chapter lists all functions of NC series controller. Users can quickly view all functions from the table of group menu.

4

4.1 Table of system group menu

For all modes

POS coordinates function			
Layer 1	Layer 2	Layer 3	Layer 4
ABS	-	-	-
REL (Connect to the physical axis to display the axial clear function)	CLR ALL	-	-
	CLR X	-	-
	CLR Y	-	-
	CLR Z	-	-
	CLR A	-	-
	CLR B	-	-
	CLR C	-	-
MECH	-	-	-

Program edit mode

PRG program function file manager			
Layer 1	Layer 2	Layer 3	Layer 4
COPY FILE	-	-	-
PASTE	-	-	-
DEL (file/folder)	-	-	-
SEL TOGL/CANCEL	-	-	-
CANCEL ALL	-	-	-
SEL ALL	-	-	-
SEQUENCE	NAME	-	-
	SIZE	-	-
	DATE	-	-
NEW FILE	-	-	-
FOLDER	-	-	-
RENAME	-	-	-
FIND FILE	-	-	-
MERGE	-	-	-
MACRO	-	-	-
DXF	-	-	-
GRAPHIC EDIT (for lathe system)			

Program edit mode

PRG program function - file editor			
Layer 1	Layer 2	Layer 3	Layer 4
File editing	COPY	-	-
	PASTE	-	-
	DEL	-	-
	UNDO	-	-
	B START	-	-
	B END	-	-
	LABLE	-	-
	STRING	NEXT	-
		PREV	-
		REPLACE	-
		REPLACE ALL	-

Auto mode

Layer 1	Layer 2	Layer 3	Layer 4
SF set	-	-	-
START	RUN	-	-
FILE SCAN	LOAD	-	-
	CLR	-	-
	CLR ALL	-	-

JOG/MPG feeding mode - program editing

Layer 1	Layer 2	Layer 3	Layer 4
SF set	-	-	-
TEACH	RAPID	-	-
	LINEAR	-	-
	ARC	P1	-
		P2	-
		P3	-
		PLANE SEL	-
	DEL	-	-
	SAVE	-	-
	NEW FILE	-	-
	MECH /ABS	-	-

Manual input mode-program editing

Layer 1	Layer 2	Layer 3	Layer 4
LOAD	-	-	-
SAVE	-	-	-
CLEAR	-	-	-

Homing mode-program editing

Layer 1	Layer 2	Layer 3	Layer 4
SF set	-	-	-

For all modes

Offset (OFS) function			
Layer 1	Layer 2	Layer 3	Layer 4
COORD	AUTO	CLR REL	-
		CLR ALL	-
		SEL L	-
		SET L/2	1^{st} point
			2^{nd} point
			SET
		SET P	-
	ABS	-	-
	INC	-	-
	SQUARE	X1	-
		X2	-
		Y1	-
		Y2	-
		SET	-
		SET Z	-
	CIRCLE	P1	-
		P2	-
		P3	-
		SET	-
		SET Z	-
CUTTER (for milling system)	ABS	-	-
	INC	-	-
	H SET	-	-
	CLEAR	H/D	-
		WAER	-
		LIFE	-

Offset (OFS) function			
Layer 1	Layer 2	Layer 3	Layer 4
		ALL	-
CUTTER (for lathe system)	LENGTH	ABS	-
		INC	-
		CLR ALL	-
		AX CLR	-
		LENGTH OFS	-
	WEAR	ABS	-
		INC	-
		CLR ALL	-
		AX CLR	-
	CUTTER END	ABS	-
		INC	-
		CLR ALL	-
		AX CLR	-
MAGA	Maga 1	SET (Jog mode only)	-
		RST ALL (jog mode only)	-
		LOCK (jog mode only)	-
		UNLOCK (jog mode only)	-
	Maga 2	SET (Jog mode only)	-
		RST ALL (jog mode only)	-
		LOCK (jog mode only)	-
		UNLOCK (jog mode only)	-
MACRO	LOCAL	-	-
	GLOBAL	-	-
	HOLD	-	-
	EXPAND	-	-

For all modes

Graphic (GRA) function			
Layer 1	Layer 2	Layer 3	Layer 4
CUTTING PATH	X-Y / Y-Z / X-Z / X-Y-Z (This is not available in lathe system)	-	-
	CENTER	-	-
	ZOOM IN	-	-

4

Graphic (GRA) function			
Layer 1	Layer 2	Layer 3	Layer 4
	ZOOM OUT	-	-
	DRAW	-	-
	STOP DRAW	-	-
	UP	-	-
	DOWN	-	-
	LEFT	-	-
	RIGHT	-	-
CUTTING PREVIEW (Auto mode only)	X-Y / Y-Z / X-Z / X-Y-Z (This is not available in lathe system)	-	-
	CENTER	-	-
	ZOOM IN	-	-
	ZOOM OUT	-	-
	PREVIEW	-	-
	CANCEL PREVIEW	-	-
	UP	-	-
	DOWN	-	-
	LEFT	-	-
	RIGHT	-	-

For all modes

Alarm (ALM) function			
Layer 1	Layer 2	Layer 3	Layer 4
ALARM	-	-	-
HISTORY	CLR ALL	-	-

For all modes

Diagnosis (DGN) function			
Layer 1	Layer 2	Layer 3	Layer 4
PROCESS	SET	-	-
	CLR TIME	-	-
	CLR NR	-	-
USER VAR	USER VAR	DEL	-
		US DEC	-
		HEX	-
		S DEC	-
		FLOAT	-
	SYS VAR	-	-

Diagnosis (DGN) function			
Layer 1	Layer 2	Layer 3	Layer 4
	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	-
	EDITOR (edit mode only)	LD	-
		LDI	-
		LDP	-
		LDF	-
		OUT	-
		APP	-
		—	-
		|	-
		DEL V-LN	-
		ADD LN	-
		DEL LN	-
		DEL	-

4

Diagnosis (DGN) function			
Layer 1	Layer 2	Layer 3	Layer 4
		LABLE	-
		TABLE	-
		SYMBOL	X
			Y
			M
			A
			T
			C
			D
			P
			I
			DEL
			COPY
			PASTE
		SAVE	-
		IMPORT	IMPORT
		EXPORT	EXPORT
			NEW FILE
		JUMP TO	-
		SELECT	-
		CUT	-
		COPY	-
		PASTE	-
	SET (edit mode only)	ON	-
		OFF	-
		RUN/STOP	-
	JUMP TO	-	-
SYS MONI	SRV MONI	-	-
	I/O MONI	-	-
	VAR MONI	SYS VAR	-
		CH VAR	-
		AXIS VAR	-
		IF VAR	-
		MLC VAR	-
		US DEC	-
		BIN	-
		HEX	-

Diagnosis (DGN) function			
Layer 1	Layer 2	Layer 3	Layer 4
		S DEC	-
SYSTEM	STATUS	-	-
	FW SN	-	-
	HW SN	-	-
	M STATUS	DEL	-
PWD	S SCP	UNLOCK	-
		LOCK	-
		SYS CHECK	-
	M SCP	PWD CHG	-
		LOCK/UNLOCK	-
		RST U1	-
		RST U2	-
		ENABLE	OK
			CANCEL ALL
			DEFAULT
		RESET	-
	U1 SCP	PWD CHG	-
		LOCK/UNLOCK	-
	U2 SCP	PWD CHG	-
		LOCK/UNLOCK	-
	EXPIRE	SETTING	-
		RELEASE	-
		EXP SCP	PWD CHG
			LOCK/UNLOCK
TUNNING (Jog or MPG mode only)	NEXT AX	-	-
	READ	-	-
	COMPUTE	-	-
	WR GAIN	-	-
	WR NOTH	-	-
	RUN	-	-
TUNNING (Jog or MPG mode only)	JOG←	-	-
	JOG→	-	-
	POS 1	-	-

Diagnosis (DGN) function			
Layer 1	Layer 2	Layer 3	Layer 4
	POS 2	-	-
	TAP RIV	TAP SET	-
	SERVO	READ SRV	-
	SYN CONTROL	POS SET	-
TEXT WR	-	-	-
IMPORT	IMPORT	-	-
	SEL ALL	-	-
	CLR ALL	-	-
EXPORT	EXPORT	-	-
	SEL ALL	-	-
	CLR ALL	-	-
LOGO WR	-	-	-

For all modes

Parameter (PAR) function			
Layer 1	Layer 2	Layer 3	Layer 4
PROCESS	-	-	-
OPERATE	-	-	-
MAGA	-	-	-
SPINDLE	-	-	-
MACHINE	-	-	-
HOME	-	-	-
NETWORK	DEFAULT	-	-
COMP	OK	-	-
	um	-	-
	um+	-	-
	IMPORT	-	-
	IMPORT +	-	-
SYSTEM	DEFAULT	-	-
	COLOR	-	-
MLC	DEFAULT	-	-
	COLOR	-	-
GRAPHIC	DEFAULT	-	-
	COLOR	-	-
SERVO	READ	-	-
SEARCH	-	-	-

Parameter (PAR) function			
Layer 1	Layer 2	Layer 3	Layer 4
CONFIG (Except Auto and MDI mode)	OK	-	-
SET RIO (Except Auto and MDI mode)	OK	-	-
PAR GROUP	SAVE	-	-
	DEL GROUP	-	-
	WRT PAR	-	-
	READ PAR	-	-
	PAR SEQ	-	-
	ALLOCATE	-	-

For all modes

Software panel (SOFT) function (Example: without physical control panel)				
Control panel functions	Program execution	MPG simulation	Tool magazine forward	Spindle forward
	Stop execution	Mechanical lock	Tool magazine backward	Spindle stop
	Single step pause	Program dry run	Chip removal forward	Spindle backward
	Selection stop	Mechanical lock	Chip removal backward	-
	Single step ignore	Z-axis lock	Blow air	Spindle positioning
	Cutting fluid	Working light	Program protection	Limit remove
Factor adjust	Increasing	-	-	-
	Decreasing	-	-	-
	100%	-	-	-
	0%	-	-	-
Axis operations	X←	-	-	-
	X→	-	-	-
	Y↗	-	-	-
	Y↙	-	-	-
	Z↑	-	-	-
	Z↓	-	-	-

4

Software panel (SOFT) function (Example: with physical control panel)			
Layer 1	Layer 2	Layer 3	Layer 4
Control panel functions	Program dry run	Chip removal forward	-
	Function lock	Chip removal backward	-
	Z-axis lock	Auto power off	-
	Mechanical lock	Program protection	-
	Spindle positioning	User-define 1	-
	Blow air	User-define 2	-

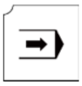
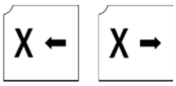
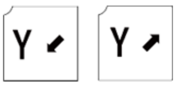
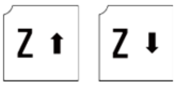
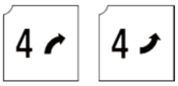
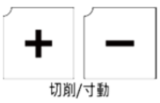
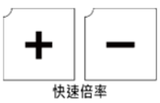
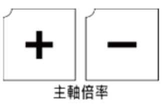
4.2 Primary control panel function keys

Name	Description	Supported Mode
POS	One of the group keys. Coordinates display group key.	All
PRG	One of the group keys. Program edit group key.	All
OFS	One of the group keys. Coordinates setup and tool offset setup group key.	All
DGN	One of the group keys. Diagnosis function, system parameter, and system status group key.	All
ALM	One of the group keys. Alarm display group key.	All
GRA	One of the group keys. Path display group key.	All
PAR	Special group key. System parameter setup group key.	All
SOFT	Special group key. Software control panel group key.	All
RESET	Reset key	All
X I ~ D H	Axis position and command code	PRG group
0 ^ ~ 9 /	Numeric key (computing symbol)	PRG, OFS, DGN group
• >	Decimal point (computing symbol)	PRG, OFS group
−]	Negative sign (computing symbol)	PRG, OFS group
▲ PAGE UP ▼ PAGE DN	Keys for page up and page down respectively	PRG, OFS, DGN group
⬆ ; ⬇ , ⬅ & ➡ #	Arrow keys (Up, Down, Left and Right) (computing symbol)	PRG, OFS, DGN group
HOME END	Jump to beginning (end) of word	PRG group

Name	Description	Supported Mode
SPACE	Space	PRG group
SHIFT	Upper/lower case shift	PRG group
DEL INS	Delete (Insert)	PRG group
BACK SPACE	Delete the letter in front of cursor	PRG group
ENTER	Enter key	PRG, OFS, DGN group
EXIT	Exit dialog box	PRG, DGN group
( )	Parentheses	PRG group
◀ ▶	Left and right function key	All
F1 ~ F6	Function keys (Some models have F7 and F8)	All

4.3 Function keys of secondary control panel

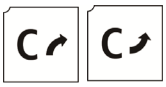
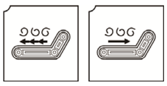
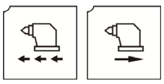
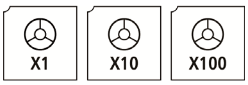
- Milling system

Name	Description
	Auto mode: For program execution
	Edit mode: For file management and program editing
	Jog mode: For manually operate the machine tool
	MPG mode: Operate machine tools axis via MPG
	Manual mode: For simple program input and execution
	Homing mode: Promptly return to the original point
	Fast positioning mode: Move axially according to the selected ratio
	X-axis forward, X-axis backward: In JOG mode, manually operate X-axis in forward or backward direction
	Y-axis forward, Y-axis backward: In JOG mode, manually operate Y-axis in forward or backward direction.
	Z-axis forward, Z-axis backward: In JOG mode, manually operate Z-axis in forward or backward direction.
	Rotation-axis forward, Rotation-axis backward: In JOG mode, manually rotate the axis in forward or backward direction.
	Spindle forward: Spindle moves forward in manual control.
	Spindle stop: Spindle stops in manual control.
	Spindle backward: Spindle moves backward in manual control.
	FEEDRATE/JOG%: Keys for increasing or decreasing the cut feeding speed and jog ratio
	RAPID% (Rapid feeding ratio): Keys for increasing or decreasing the fast feeding ratio
	SP% (Spindle factor): Keys for increasing or decreasing the spindle speed ratio.

4

Name	Description
	Single step pause: After enabling the function, the system stops execution when finish one single step.
	Limit release: When the limit protection is effective, it is the main key to clear the limit alarm.
	Single block ignore: Enter “ / ” in the front and press this key to enable this function.
	Tool magazine forward: In safe mode, it enables the tool magazine to move one position in forward direction.
	Tool magazine backward: In safe mode, it enables the tool magazine to move one position in reverse direction.
	Selection stop: Press this button and execute M01 command to enable this function.
	MPG simulation: During program execution, after this function is enabled, the MPG can be used to control the speed.
	Cutting fluid ON/OFF: The switch of switching On/Off the cutting fluid
	Working light: The switch of turning On/Off the working light

- Lathe system

Name	Description
	Rotation axis in forward and backward direction: In jog mode, rotate the rotation axis in forward or backward direction.
	Chip conveyor runs in forward / backward direction
	Tailstock center: Forward /backward
	Spindle hydraulic chuck: release / tighten
	MPG ratio selection: X1, X10, X100
CUM 1 CUM 2 CUM 3	User-defined keys

4.4 Table of function keys (For NC_EM model)

NC Key	PC Keyboard	Description
F1 ~ F8 (Function Key)	F1 ~ F8	Function keys
► (Function Key)	Tab	Next layer (function key)
◄ (Function Key)	Ctrl + Tab	Previous layer (function key)
POS	Ctrl + F1	POS group key
PRG	Ctrl + F2	PRG group key
OFS	Ctrl + F3	OFS group key
DGN	Ctrl + F4	DGN group key
ALM	Ctrl + F5	ALM group key
GRA	Ctrl + F6	GRA group key
PAR	Ctrl + F7	PAR group key
SOFT	Ctrl + F8	SOFT group key
Numeric keys	Numeric keys	-
Typewritter keys	Typewritter keys	-
Symbol	Symbol	-
Cursor control keys	Cursor control keys	-
PAGE UP / PAGE DN	Page Up / Page Down	-
BACKSPACE	Backspace	-
SPACE	Space	-
DEL / INS	Delete / Insert	-
SHIFT	Shift	-
HOME / END	Home / End	-
ENTER	Enter	-
EXIT	Esc	-
RESET	Ctrl + Esc	-
-	F12	Help (Description of each key)
SHIFT + GRA	PrtScn	Capture Screen

(This page is intentionally left blank.)

Mode Function

5

NC controller system provides multiple operating modes. This chapter will briefly introduce each mode function.

5.1 Auto mode (AUTO)

5

The system must be set to AUTO mode before a program is executed. This enables users to validate machining program, cutting conditions, and coordinates of positions before execution as well as to avoid unexpected operation by incorrectly pressing keys in non-auto mode. In this mode, only executing program file is allowed; functions such as program editing and manually operating the axial movement are not available here.

5.2 Program edit mode (EDIT)

Program editing only can be done in EDIT mode. In EDIT mode, users may access various program editing functions available in PRG group. Please note that program execution and limiting axial operating direction are not allowed.

5.3 Manual input mode (MDI)

Users can input a single block program in the screens of PRG group and execute it in MDI mode. As most MDI programs are simple ones manually entered by users, there is no need to have too much program content. MDI's PRG group screens allow a single block program of up to 17 statements. Functions of program editing, program execution or manually operating axis directions are not available in this mode.

5.4 MPG feeding mode (MPG)

In MPG mode, it allows users to manually control the axis via external MPG. Users are able to manually control the moving direction of each axis more promptly and accurately. Functions such as program editing, program execution, and jog operation are not available in this mode.

5.5 Jog feeding mode (JOG)

Pressing relevant axial movement keys in secondary control panel can carry out axial jog offset in JOG mode. The speed and distance of each jog movement is controlled by the jog factor key. The workbench can be moved with the rapid feeding activation key and axial keys. The axial moving speed is set by the rapid factor and can enable moving the workbench in long distance of each axis. Both program execution and editing functions are unavailable in JOG mode. It allows the axial keys in secondary control panel to carry out axial movement.

5.6 Homing mode (HOME)

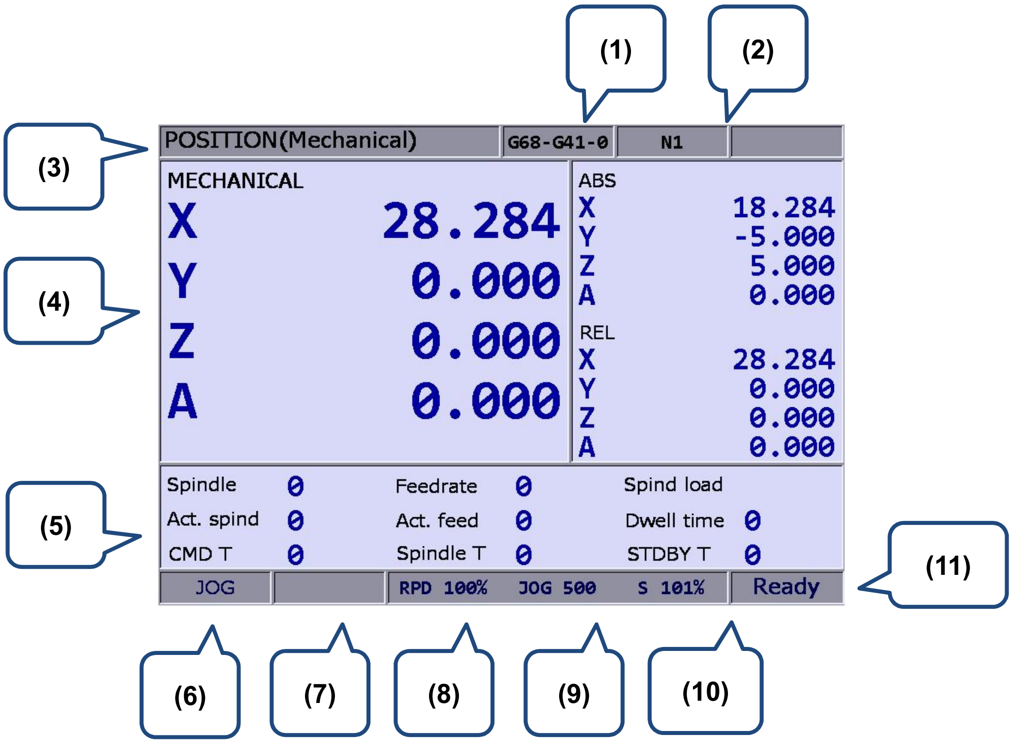
The HOME mode simplifies the manual origin reset operation. When the system is set to Home mode, users can use axial movement keys on the secondary control panel and the axis will return to its mechanical origin. After re-starting the controller, it is required to conduct homing procedure first to make each axis return to the origin. When homing completed, program can then be executed. Otherwise, the controller stops the program execution function.

5

5.7 Group screen overview

Screens of function groups of this controller provide a full range of information. Some of the screens of each group are illustrated below.

POS group:

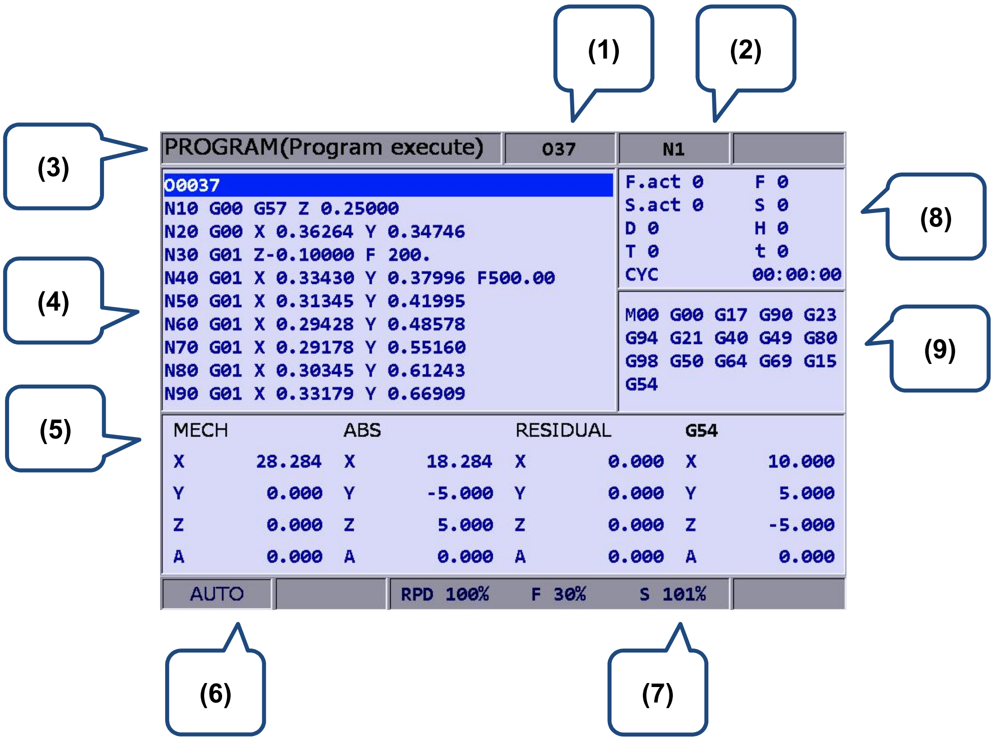


(1) Name of current program
(2) Line being executed
(3) Display of current group
(4) Current coordinates data
(5) S: Spindle speed (command value)
F: Feed rate (command value)
S.lod: Spindle load
S.act: Actual spindle speed rate
F.act: Actual feed rate
T: Pause time
T: Tool number
T.spindle: Spindle number
T.stdby: Standby tool number

(6) Current system mode
(7) Alarm display
(8) Fast speed ratio
(9) Feeding factor ratio
(10) Spindle factor ratio
(11) Current status

As shown in the figure above, the system status column tells the status of this system for the controller user's reference. Valid statuses of the system in terms of priority are: MLC stop, servo not ready, emergency stop, in process, in operation, program stop, and preparation completed.

PRG group (auto mode):



(1) Name of current program

(2) Line being executed

(3) Display of current group

(4) Program content being executed

(5) Information of each coordinate that has being executed

(6) Current system mode

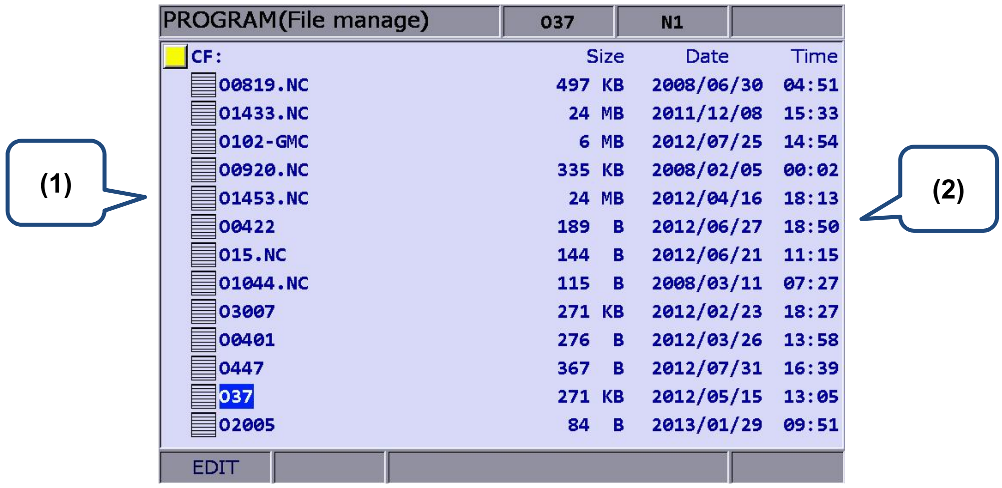
(7) Each current motion rate

(8) F.act: Actual feed rate
S.act: Actual spindle speed rate
D: Tool radius compensation ID
H: Tool length compensation ID
T: Tool number
F: Feed rate
S: Spindle speed
T: Pause time
CYC: Single processing time

(9) Current command status

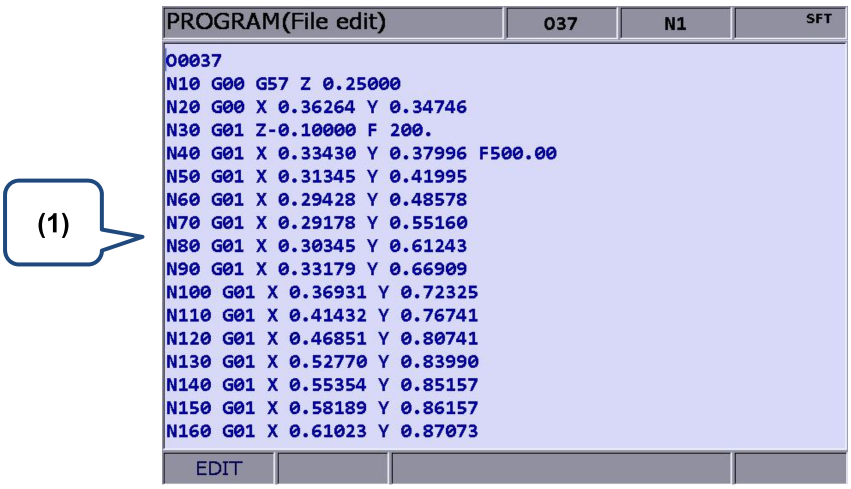
5

Program edit mode:



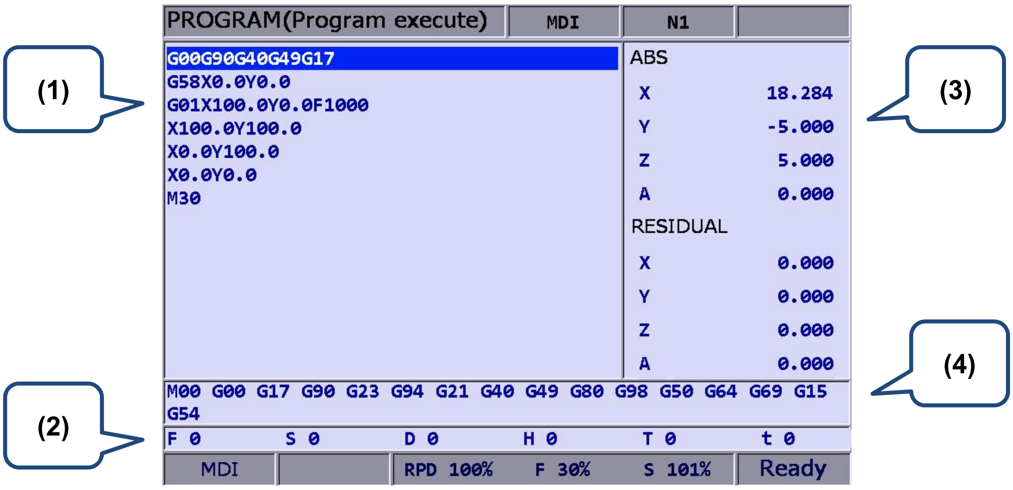
(1) File list: Display folder and program files

(2) File information: Display data by file size or folder modification date and time.



(1) File content: Display program statements contained in the file

Manual input mode:



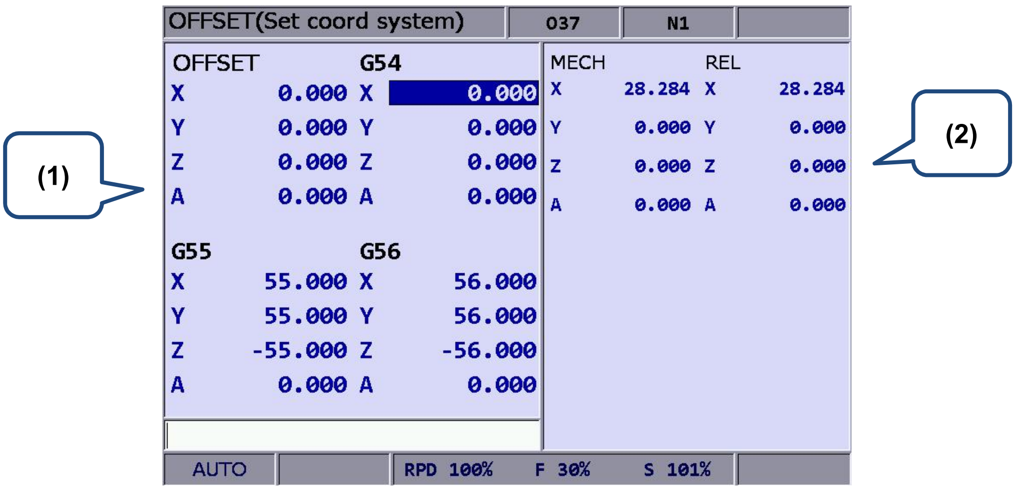
(1) Manual mode

(2) The information of feed rate, spindle speed and compensation

(3) Coordinates information: Display the information of absolute / remaining coordinate

(4) Command status

OFS group (coordinates system data)

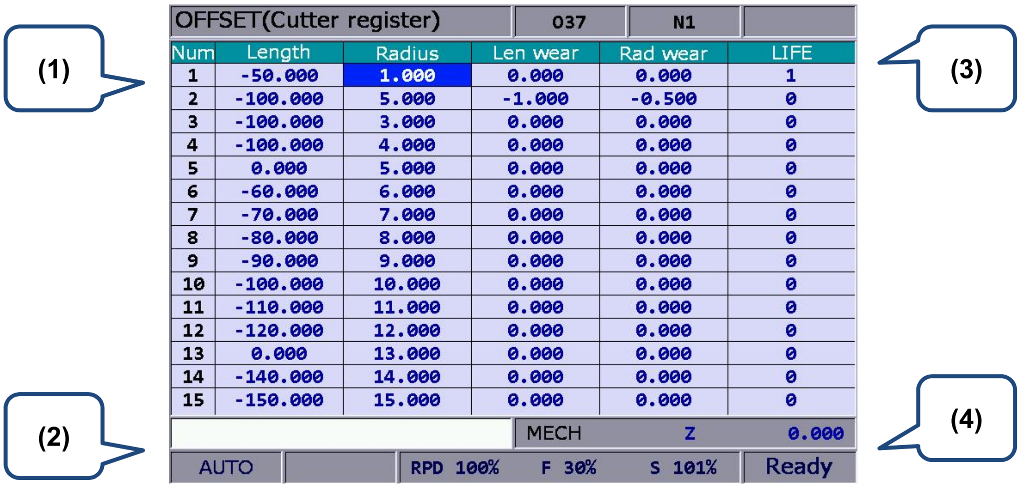


(1) Coordinate system setup: Offset coordinates / G54 ~ G59 coordinates

(2) Coordinate information: Mechanical / Relative coordinate

5

Tool data:



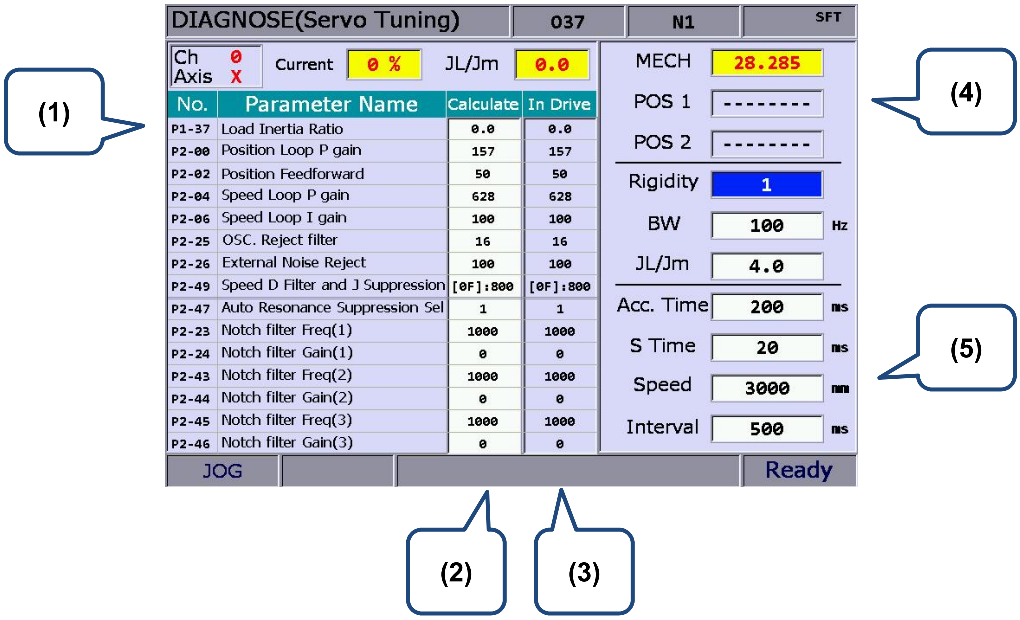
(1) Compensation ID (H/D)

(2) Compensation data input column

(3) Compensation data: Tool length, tool radius, length, and radius compensation

(4) Auxiliary display: Display current mechanical coordinates and actual position of Z-axis

DGN group (Gain adjustment):



(1) Servo parameter ID: Servo parameter ID and name

(2) Calculate after gain adjustment: Display the calculation result after auto tuning

(3) System setting: Display the setting value that servo system currently applied

(4) Positioning point setting: positioning point 1 / positioning point 2

(5) Tuning conditions

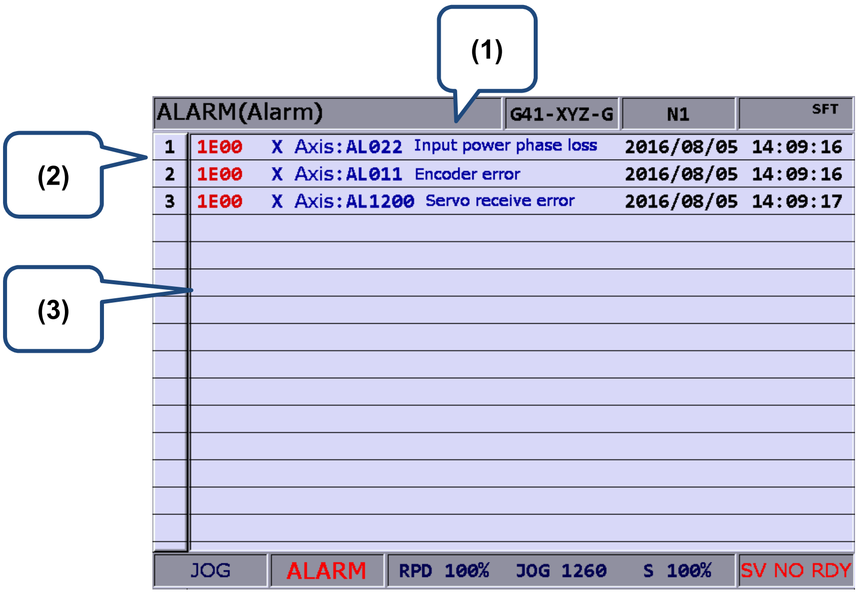
5

MLC operation/edit:

(1) MLC program

(2) Input column for command name

ALM group:



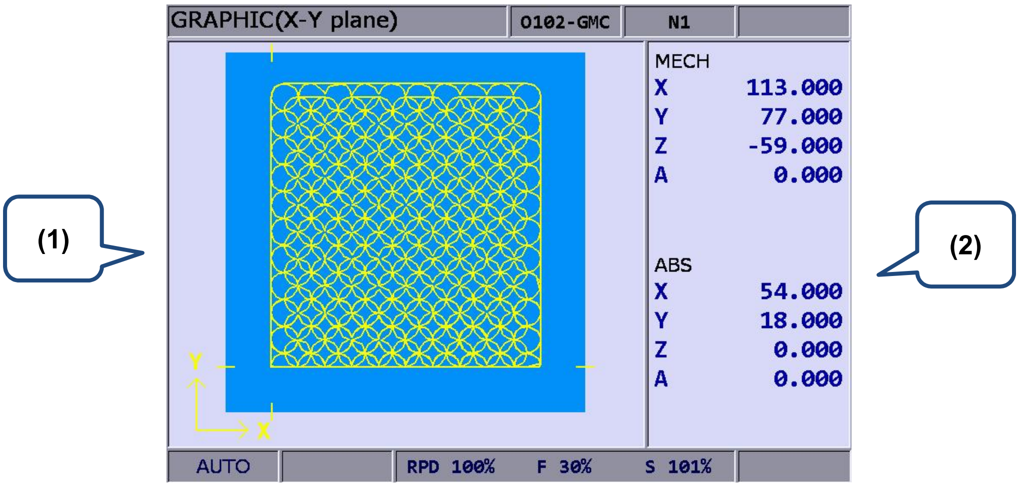
(1) Alarm message

(2) Sequence of alarm occurrence

(3) Alarm number

5

GRA group:



(1) Path diagram: Display program path

(2) Coordinate information: Mechanical coordinate / Absolute coordinate

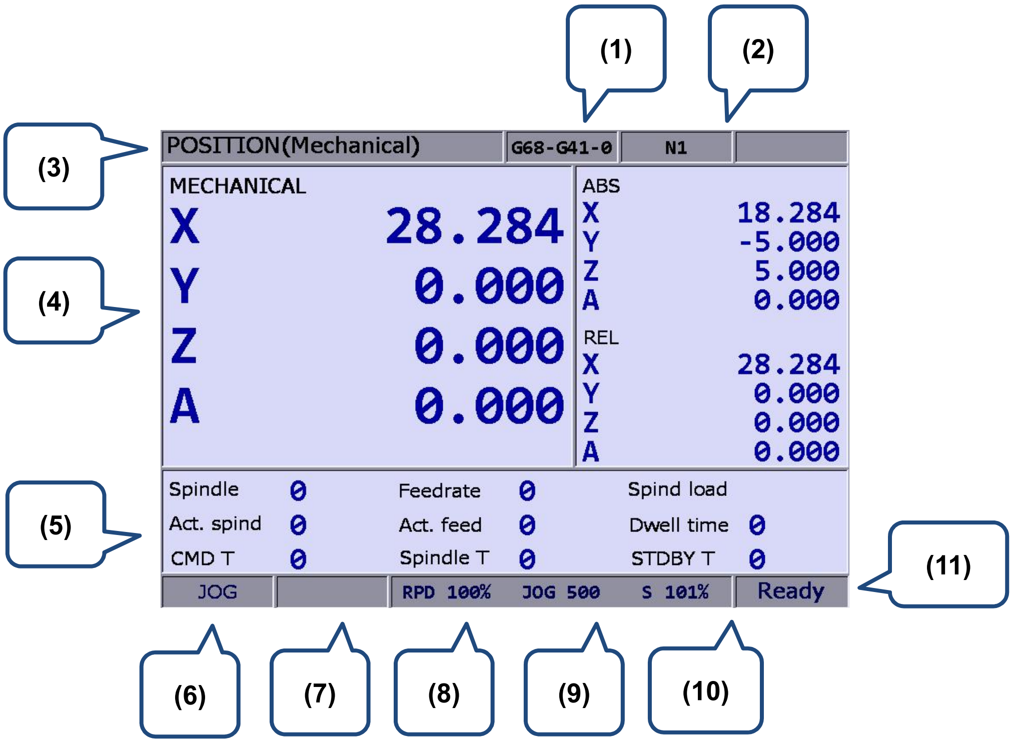
Position (POS) Group 6

POS Group presents different types of coordinates data. It includes mechanical coordinates, absolute coordinates and relative coordinates.

6

POS Group presents different types of coordinates data. It includes mechanical coordinates, absolute coordinates and relative coordinates. According to the application, up to three linear axes and one rotary axis can be added.

Note: Here we use **Framed Text** to indicate the keys in primary control panel. And **boldface letter** is used for indicating the function key.



(1) Name of current program
(2) Line being executed
(3) Current group
(4) Current coordinates data
(5) S: Spindle speed (command value)
F: Feed rate (command value)
S.lod: Spindle load
S.act: Actual spindle speed rate
F.act: Actual feed rate
t: Pause time
T: Tool number
T.spindle: Spindle ID
T.stdby: Standby tool ID

(6) Current system mode
(7) Alarm display
(8) Fast speed ratio
(9) Feeding factor ratio
(10) Spindle factor ratio
(11) Current status

6

6.1 Absolute coordinates

The absolute coordinate value is displayed based on the origin of the G code. Coordinate values are used to validate the movement position of a single block. See below for operation details.

1. Press the **POS** key to enable the display of coordinate group function, namely absolute coordinates, relative coordinates, and mechanical coordinate options on the function bar.

2. Press the **ABS** key to enter the absolute coordinates screen.

6.2 Relative coordinates

The relative coordinates indicate the moving distance from the origin. See below for operation details.

1. Press the **POS** key to enable the display of coordinate group function, namely absolute coordinates, relative coordinates, and mechanical coordinate options on the function bar.

2. Press the **REL** key to enter the relative coordinates screen.

3. Press the **CLR** key in the lower layer function bar to clear relative coordinate values of all axes.
   Press the **CLR X** key to clear the relative coordinate value of the X-axis.
   Press the **CLR Y** key to clear the relative coordinate value of the Y-axis.
   Press the **CLR Z** key to clear the relative coordinate value of the Z-axis.
   Press the **CLR A** key to clear the relative coordinate value of A-axis.
   Press the **CLR B** key to clear the relative coordinate value of B-axis.
   Or press the **CLR C** key of the next page to clear the value shown on relative coordinates of the C-axis.

Note: Clear function for X-, Y-, Z-, A-, B-, and C-axis is displayed only when they are set to correspond to actual axes.

6

6.3 Mechanical coordinates

The mechanical coordinate data is defined based on the real mechanism. This data is unchangeable and cannot be cleared. And this data does not vary with the selected workpiece coordinates. See below for operation details.

1. Press the **POS** key to enable the display of coordinate group function, namely absolute coordinates, relative coordinates, and mechanical coordinate options on the function bar.

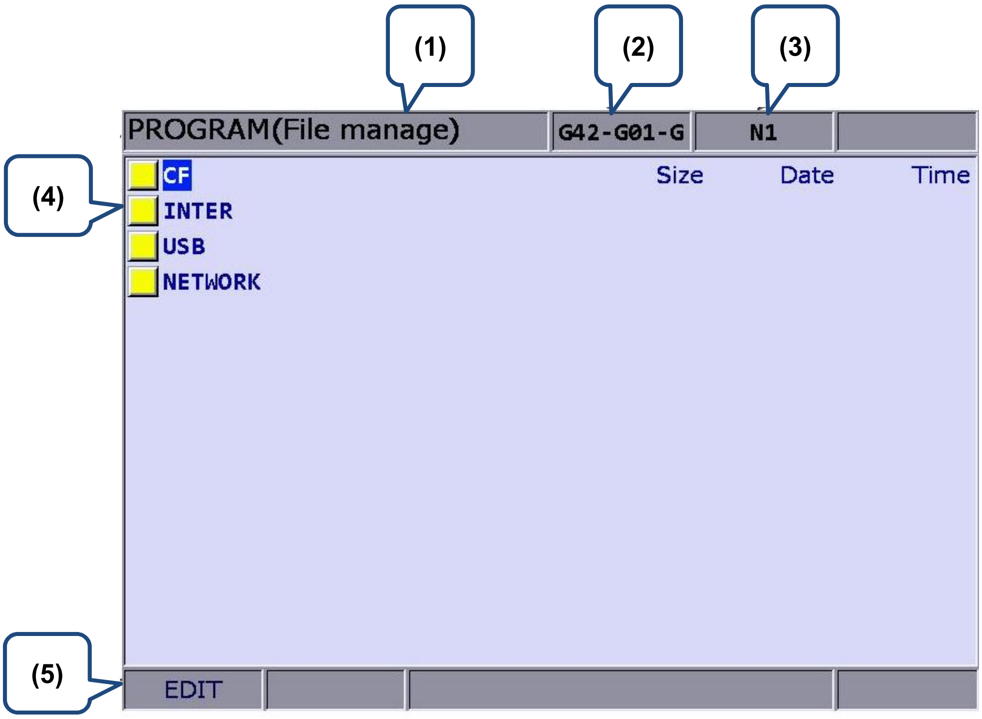
2. Press the **MECH** key to enter the mechanical coordinates screen.

Program (PRG) Group 7

PRG Group function allows you to manage and edit G code and Macro files. And it also provides functions for different operation modes.

The PRG group manages and edits G code and macro files. The function of [File manage] is divided into three sections: (1) CF Card, internal memory, USB drive, and network; (2) folders and G code files; (3) G code files only. Each section has its exclusive functions, e.g. breakpoint search function in auto mode and program editing and execution in manual mode.

Note: Here we use **Framed Text** to indicate the keys in primary control panel. And **boldface letter** is used for indicating the function key.



(1) Current group
(2) Current program
(3) The line being executed
(4) Disk option
(5) Current mode

Set the system to **Edit mode**. Then, press the **PRG** key in the primary control panel to enter the screen of [Program].

Then, press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor in the screen of [File manage] and press the **ENTER** key to enter the second or third layer to select G code files. Next, select the desired G code file and press the **ENTER** key to enter file editing screen. You can use ↑ and ↓ keys to scroll one line and use **PAGE UP** and **PAGE DN** keys to scroll 20 lines at a time to show the file content.

Note: See the table below for the recommended format of the USB drive.

USB drive specifications	
Format	FAT32
Capacity	As required

7.1 Network Setting

The NETWORK function remotely connects to a PC through Ethernet. With CNC Network software, this remote connection enables one PC to do online file management with multiple NC controllers. Users can share files with PCs, and do file management and transmission-along-with-machining (DNC).

Please set up the communication protocol between the NC system and PC before using the networking function. See below for setup steps:

NC system communication protocol: Screen of PRA group > Network Setting.

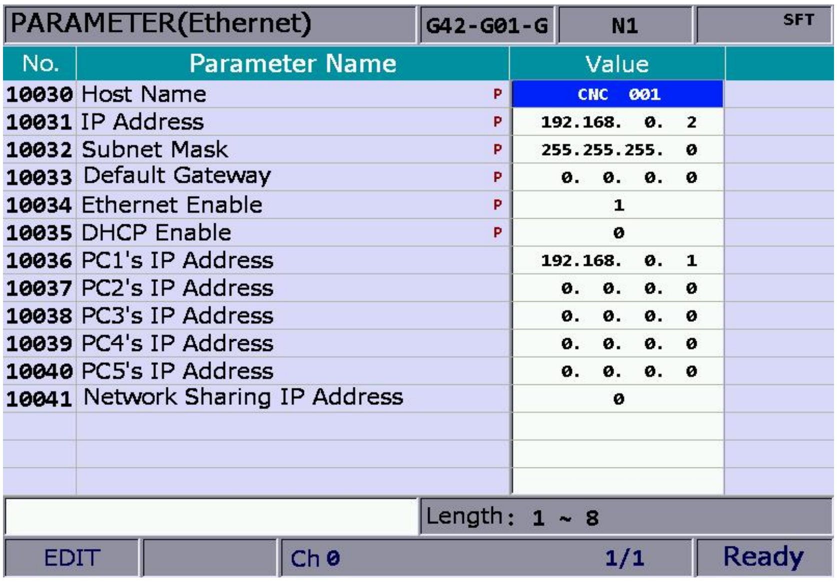


Figure 7.1.1

Network setup parameters		
Code	Name	Range or Formats
10030	Host name	Length: 1~ 8 Actual setting: 1~ 8 characters
10031	IP address	Length: ××× · ××× · ××× · ××× Actual setting: 192 · 168 · 0 · 2
10032	Subnet mask	Length: ××× · ××× · ××× · ××× Actual setting: 255 · 255 · 255 · 0
10033	Default gateway	Length: ××× · ××× · ××× · ××× Actual setting: 0 · 0 · 0 · 0
10034	Network function ON	Length: 0 ~ 1 Actual setting: 1
10035	DHCP ON	Length: 0 ~ 1 Actual setting: 0
10036	IP address of remote computer 1	Length: ××× · ××× · ××× · ××× Actual setting: 192 · 168 · 0 · 1
10037	IP address of remote computer 2	Length: ××× · ××× · ××× · ××× Actual setting: 0 · 0 · 0 · 0
10038	IP address of remote computer 3	Length: ××× · ××× · ××× · ××× Actual setting: 0 · 0 · 0 · 0
10039	IP address of remote	Length: ××× · ××× · ××× · ×××

Network setup parameters		
Code	Name	Range or Formats
	computer 4	Actual setting: 0 · 0 · 0 · 0
10040	IP address of remote computer 5	Length: ××× · ××× · ××× · ××× Actual setting: 0 · 0 · 0 · 0
10041	IP address of remote folder sharing	Length: 0 ~ 5 Actual setting: 0

Communication protocol of PC: Set up Internet Protocol (TCP/IP) Properties in PC operating system (see Figure 7.1.2) or **CNC Network software** > **Options**

Network setup in PC operating system:

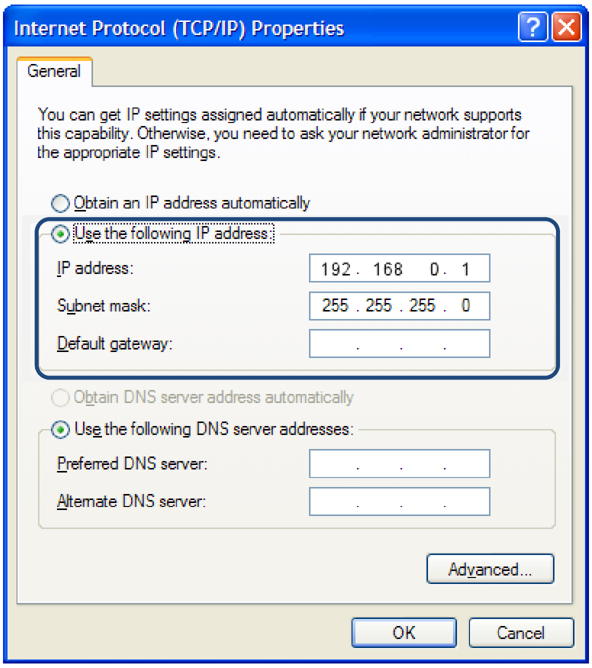


Figure 7.1.2

Steps:

1. Select **Use the following IP address** and then enter the address below:

   IP address: **192 . 168 . 0 . 1**
   Subnet mask: **255 . 255 . 255 . 0**

2. Press **OK** to complete the setting.

Network setup for Network software:

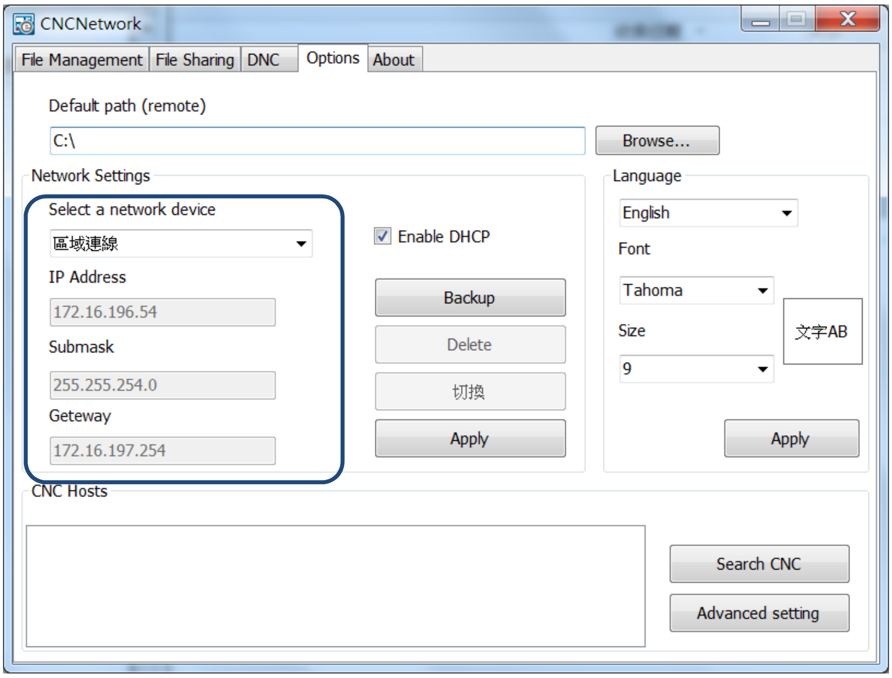


Figure 7.1.3

Steps:

1. Start the **CNC Network software**. Enter the [Options] screen and set up as below:

   IP address: **192 . 168 . 0 . 1**
   Subnet mask: **255 . 255 . 255 . 0**

2. After entering the address, press **Search CNC** to connect with the CNC based on the settings given here.

7

DNC Connection:

Through Network software, users may open the shared files in file sharing list. Then, execute G code in transmission-along-with-machining (DNC) mode via Ethernet.

No extra disk space is required for file storage as only the path of shared files is recorded.

See the operation steps described below:

1. Complete the Ethernet communication setting for connection between PC and NC system.
2. Start the **CNC Network software**.
3. Click on the Function bar > **DNC**.

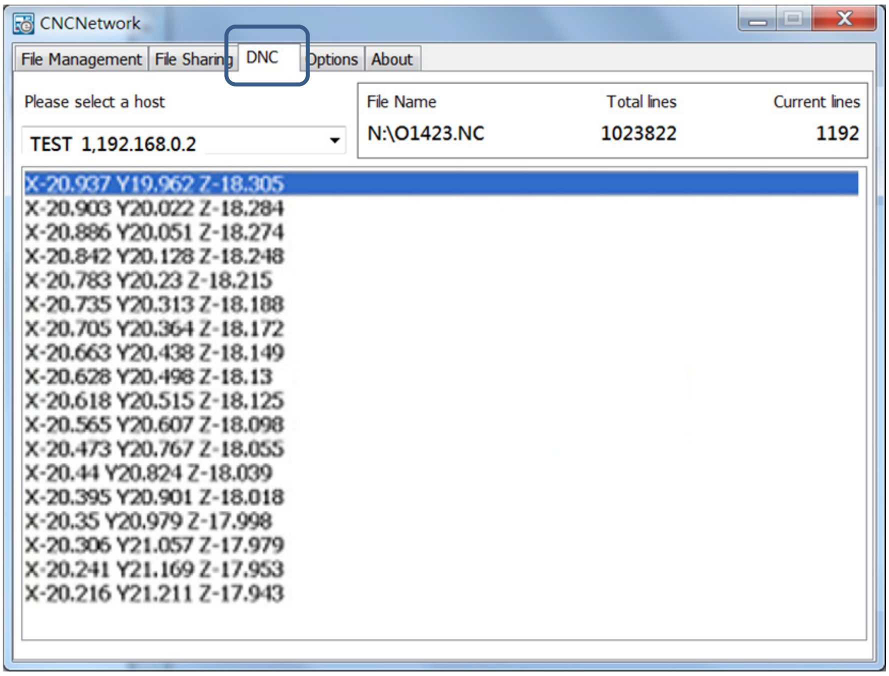


Figure 7.1.4

4. Enter the **Edit mode** of NC system and then enter the top layer NETWORK\Option in [File manage].

Figure 7.1.5

After the shared file is displayed, select and open the G code file that has been set to be shared.

5. Set the system to **Auto mode** and then press **Cycle start** to start running the G code file with DNC connection. The execution method is the same as that of the general file.

6. During DNC execution, file information can be displayed in the window of **DNC** provided by CNC Network software. The information includes name of connected system, name of running DNC file, total number of lines, executing line number and file contents. (File contents scroll down along with the execution progress as shown in Figure 7.1.6).

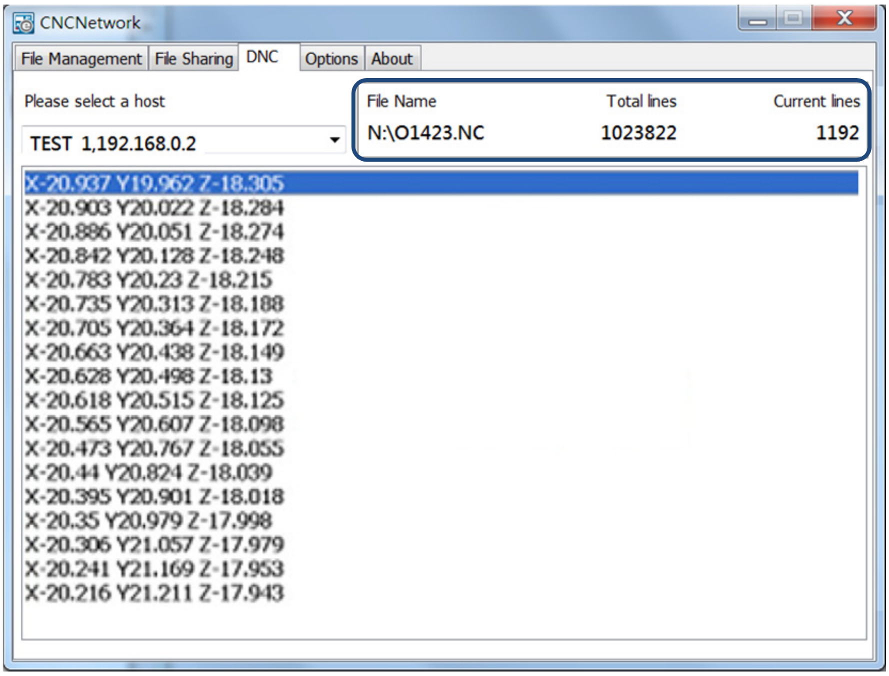


Figure 7.1.6

7.2 Create new file

Users may create a new G code file in **EDIT Mode** from the controller interface.
See the operation steps below:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to the screen of [PROGRAM].
3. In the screen of [File manage], press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor to the destination of the disk for file creation (e.g. the 2nd or the 3rd layer in CF or USB directory).
4. Press the ▶ key to display the function on next page.
5. Press the **NEW file** and the dialog box for file name will pop up.
6. Type alphanumeric letters (symbols are not included) in the box and press the **ENTER** key to create a new file.

Format of file name:

File format specifications	
Format of machining file name (G code)	No restriction on format of master file name (file name must be unique in one directory)
	O + 0001 ~ 8999 (for subrountines)
Format of macro file name (O Macro)	O + 9000 ~ 9999
Remarks in file name	Suffix a '-' symbol in the file name along with more alphanumeric letters
Allowable format for filename extension	.NC .ANC .CNC .PIM .TAP .PTP .UOO .DEMO
Format of M macro file name	M + 10000 ~ 29999
Format of G macro file name	G + 30000 ~ 49999
Maximum length of file name	31 (characters)
Storage location	Second and third management layer
Restriction symbol in file name	* / \	< > ? “ :

Note:

1. File name must be unique in one directory, e.g. O0001 and O1 are regarded as the same.
2. Only the machining files are displayed in the screen of [File manage]. The macro files display only upon special permission.
3. G code file name may include multiple decimal and the last decimal shall follow the naming rule; e.g. 1.1.1.1**.NC**

7.3 Copy

This function enables users to copy existing files in the disk drive.

See the operation steps below:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to the screen of [PROGRAM].
3. In the screen of [File manage], press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor to the destination of the disk for file creation (e.g. the 2nd or the 3rd layer in CF or USB directory).
4. Move the cursor to the target file to be copied.
5. Press the **Copy file** to copy the file. Please note that it is required to execute the paste function to create a target file.

7.4 Paste

As described in Section **7.3**, it is required to execute this function together with the copy function to copy a file. This function is one of the management functions of PRG Group.

See the operation steps below (continued from Section **7.3**).

6. Use ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor to the disk, data directory or layer of the target file.
7. Enter the directory of the target file and press the **Paste**. Then, enter a new name or use the old name of the target file in the popup dialog box. Press the **ENTER** key and file coping and pasting is done.

Note:

1. Please note that if the newly copied file exists in the same directory, then its name must differ from the source one.
2. The system prompts an information box with the message “Please copy a file at first” if no copy action has been done beforehand. The file paste function has no effect.
3. Files in the USB disk can be copied and pasted to CF card with the steps described above.

7.5 Delete (For files and folders)

This function deletes files and folders at the second layer of [File manage].

See the operation steps below:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to the screen of [PROGRAM].
3. Press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to enter the disk location and data layer of the file or folder to be deleted.
4. Move the cursor to the folder or file to be deleted.
5. Press the **DEL** and the "Do you really want to delete?" dialog box will pop up. Press "Y" and the **ENTER** key to delete the selected file or folder.

Note: The deleted file cannot be recovered by undoing the delete operation.

7.6 Select /Deselect multiple files

In addition to single file operation, users may use the select/deselect function key from the function bar in [File manage] screen to select/deselect multiple files for copying or deleting.

See the operation steps below for copying and pasting multiple files:

1. Set system to **EDIT mode**.
2. Press the **PRG** key to switch to the screen of [PROGRAM].
3. Enter the file directory for selecting multiple files.
4. In the screen of [File manage], use ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor to the desired files. Press the **SEL TOGL** to select or deselect the file (see Figure 5.6.1). Or, press the **SEL ALL**, all files will be selected. For files that have been selected, pressing the **CANCELALL** will cancel their selection.

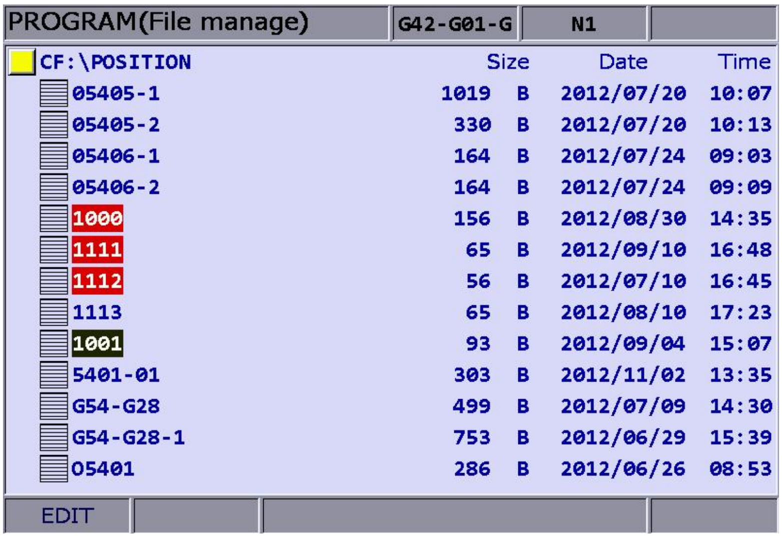


Figure 7.6.1

5. Press the **Copy file** to copy multiple files.

6. Move the cursor to another directory. Press the **Paste** to paste multiple files as shown in Figure 7.6.2

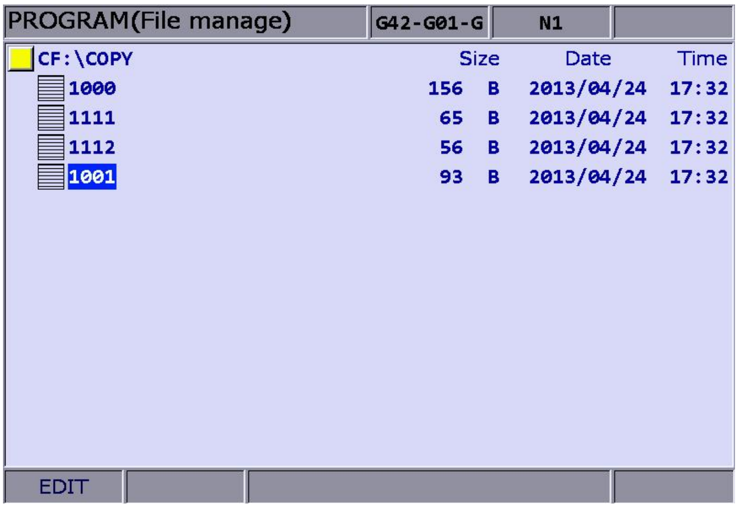


Figure 7.6.2

See the operation steps described below for deleting multiple files:

1. Set the system to **EDIT mode**.

2. Press the **PRG** key to switch to the screen of [PROGRAM].

3. Enter the file directory for selecting multiple files.

4. In the screen of [File manage], use ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor to the desired files. Press the **SEL TOGL** to select the file. For files that have been selected, pressing the **SEL TOGL** again will cancel the selection.

5. Press the **DEL** and the dialog box for confirmation will pop up (see Figure 7.6.3). Press "Y" and the **ENTER** key to delete the selected files.

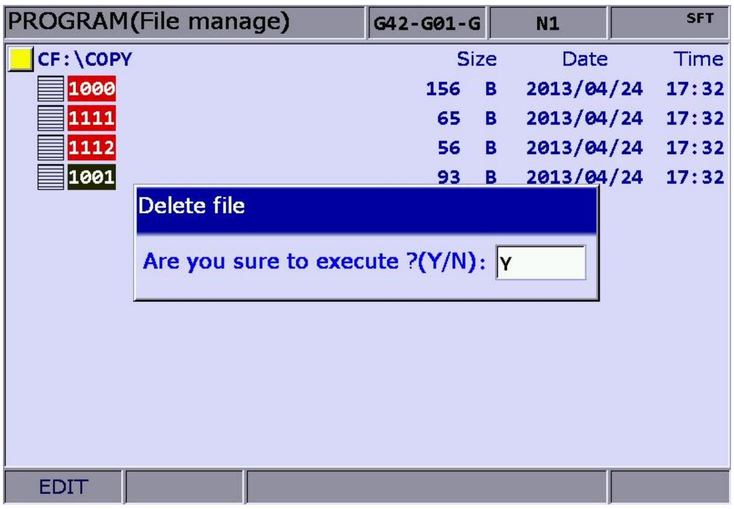


Figure 7.6.3

Note:

1. After copying multiple files in one directory, users shall paste them to another folder, which the path has to be different. If trying to copy multiple files in the same folder, the system prompts users to select another destination path and ignore the pasting operation.
2. When there are duplicated file names while copying multiple files, the NC numerical control system prompts users with an overwrite option dialog box. Users can select "Y" (yes) to overwrite the existing file, or select "N" (no) or press **EXIT** key to ignore the pasting operation.

7.7 Rename

Use this function to change the name of existing files.

See the operation steps below:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to the screen of [PROGRAM].
3. In the screen of [File manage], press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor to the destination (e.g. the 2^{nd} or 3^{rd} layer in the CF or USB directory).
4. Press the ► key to switch to the function bar at next page.
5. Move the cursor to the file that you wish to rename. Then, press the **Rename** and the dialog box for file name input will pop up.
6. Enter a new name of the file which differs from any file in the directory and press the **ENTER** key.

Note:.

1. A G code file can be added in layer two or three but not layer one in [File manage].
2. The naming format for renaming and naming for new file follows the same rule. If user enters a name that already exists in the directory, an error message will pop up and the renaming will be invalid.

7.8 Create directory

New directory for G code files can be created in the second manage layer in [File manage]. That is, the second layer of [File manage] may contain both directory folders and G code files.
See the operation steps below:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to the screen of [PROGRAM].
3. Press the ▶ key to switch to the function bar at next page.
4. Press the **FOLDER** in the second layer of the [File manage], the dialog box for entering directory name will pop up.

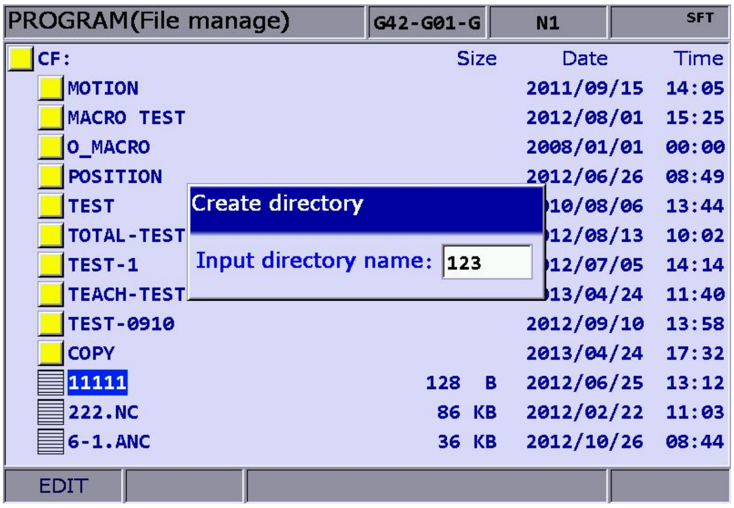


Figure 7.8.1

5. Enter the directory name and press the **ENTER** key to complete the creation.

By following the steps above, you can create a new directory in the second layer of [File manage]. Now, you can create files and edit files (such as G code files) at the third layer of [File manage].

Naming rules of directory name:

Naming rules of directory name	
Format	Any alphanumeric letter
Maximum length	31 (characters)
Storage location	The 2nd management layer

7.9 File searching

This function enables users search among many files and open a desired G code file. With a given file name users can search and open files quickly.

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. In the screen of [File manage], press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to go to the destination in the second or third layer in the disk.
4. Press the ► key to switch to the function bar at next page.
5. Press the **FIND FILE** and the dialog box will pop up. Enter the desired file name in the box and press the **ENTER** key to search and open the target file.

Note:

1. Instead of searching all directories, this file searching function is limited to one directory.
2. To find the correct file and open it, please enter the complete and exact file name you wish to search for.

7.10 File merge

This function copies and merges two G code files into one.

See the operation steps described below:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. In the screen of [File manage], press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to go to the destination in the second or third layer in the disk.
4. Select the G code file to be copied.
5. Press the **COPY FILE** to save the file in the system's clipboard.
6. Move the cursor to the directory of the target file to be merged.
7. Press the ► key to switch to the function bar at next page.
8. Press the **MERGE** and the dialog box will pop up. Enter the desired file name and press the **ENTER** key to open the target file.
9. Move the cursor to the location in the target file to paste the source file. Press the **Paste** and the dialog box will pop up. Press "Y" and enter the **ENTER** key to merge both files.
10. Execute auto save, either by switching mode, opening other files or pressing the **RESET** key, to complete the merge operation.

7.11 Sequencing

Users can arrange the sequence of directory and files by applying this function. This brings convenience when searching or managing files.

1. Set the system to "EDIT mode".
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. In the screen of [File manage], press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to go to the destination in the second or third layer in the disk.
4. Press the ► key to switch to the function bar at next page.
5. Press the **SEQUENCE** to display the function bar to the second row of the function page.
6. Press the **NAME** and then the directory and file will be displayed by the sequence of number > English (from top to bottom). Press the **NAME** again, the displayed sequence will be English > number (from top to bottom).
7. Press the **SIZE** and the file displayed sequence will start from small > large (from top to bottom). Press the **SIZE**, the sequence will be large > small (from top to bottom).
8. Press the **DATE** and the then the directory and file will be displayed by the sequence of most recent > earlier (from top to bottom). Press the **DATE** again, the sequence will be earlier > most recent (from top to bottom).

7.12 Convert DXF files

This is the interface of file manager for DXF file. Users can select the DXF file first. Then, enter the parameters values to convert the DXF file to the G code file.

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. Consecutively press the ▶ key to switch the function bar to the third row of the function page in this layer.
4. Press the **DXF** to display the interface of DXF file explorer.
5. In the screen of [File manage], press ⬆, ⬇, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to select the DXF file to be converted.
6. Then, a dialogue box will pop up and ask users to enter the value. See the figure below.

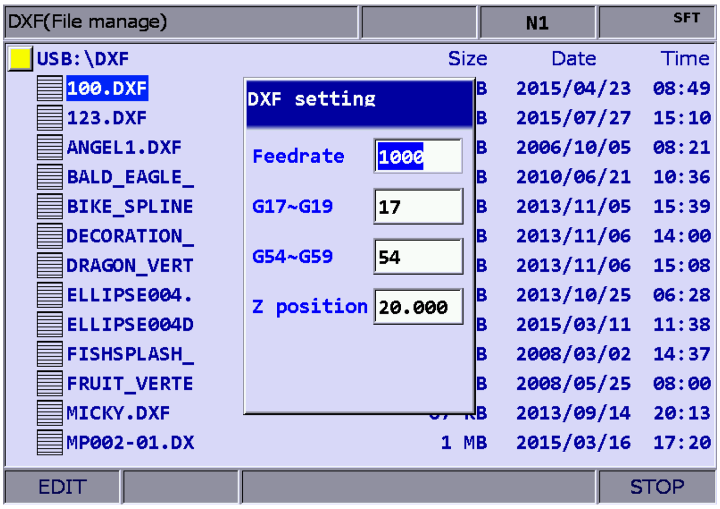


Figure 7.12.1

7. When you complete the setting of parameters values, press the **ENTER** key and an input box of "Enter new filename" will pop up.
8. Then, press the **ENTER** key to convert DXF file and G code file is stored in CF directory.
9. Now, users can execute the G code file that is just converted from DXF file.

7.13 Macro file

This function manages and edits equipment-specific macro files. With the permission, users can use all the edit functions to manage and edit macro files as described in Section 7.14. Otherwise, users can only browse existing macro files but cannot view or edit the contents. Please contact the local service provider for permission for macro file editing.

7

7.14 File editing

The edit group function enables users to edit and manage G code files. After a G code file is opened in the screen of [File manage], it will switch to the page for file editing. Move the cursor to any location in the file and then use the text, number, or edit keys in primary control panel to edit as required. After the editing is completed, switching mode, pressing the **RESET** key or open other files will automatically save the file.

See the steps described below for file editing:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. Press ⇧, ⇩, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to go to the destination in the second or third layer in the disk.
4. Select the desired G code file and press the **ENTER** key to open the file and enter edit mode.
5. Use ⇧, ⇩, ⇦, and ⇨ keys to move the cursor to any location in the file.
6. Press the text, number, and edit keys in primary control panel to edit as required
7. Save the changes by performing auto save operations, either by switching mode, opening other files, or pressing the **RESET** key.

Specifications of edit function:

Specifications of edit function	
The maximum number of characters per line	255 (characters)
Supported mode	Edit mode
Size limit of editable files	Only the file with size less than 3 MB can be edited.

Note:

1. When managing or editing the file, the function bar for editing purpose displays only when it is in **EDIT mode**. Otherwise, the PRG group function only provides program viewing and coordinates information of currently open files.
2. Users can insert the string to “( )” in the end each block in G code file as the note. “( )” cannot be placed at the front of the command block. Or the block will be regarded as the note and be ignored.

7.14.1 Search by line number

This function enables users to search contents in running G code files by line number.
See the operation steps below:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. Press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to go to the destination in the second or third layer in the disk.
4. Select the desired G code file. Then, press the **ENTER** key to open the file and enter the screen for file editing.
5. Press the ▶ key to switch to the function bar at next page.
6. Press the **LABEL** and the dialog box for entering line number (key pad 0 ~ 9) will pop up.
7. Enter the desired line number and press the **ENTER** key. The cursor will move to the given line number and finish searching.

   Rules of searching line number:

Rules of searching line number	
Maximum length of searching string	62 (characters)
Format of searching	Actual line number of program (key pad 0 ~ 9)

7.14.2 Search by strings

This function enables users to search the program by strings. Searching results vary with the fuzziness of keywords.
See the steps described below for file editing:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. Press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to go to the destination in the second or third layer in the disk.
4. Select the desired G code file. Then, press the **ENTER** key to open the file and enter the screen for file editing.
5. Press the ▶ key to switch to the function bar at next page.

6. Press the **STRING** and the dialog box for entering the string will pop up. See the figure below.

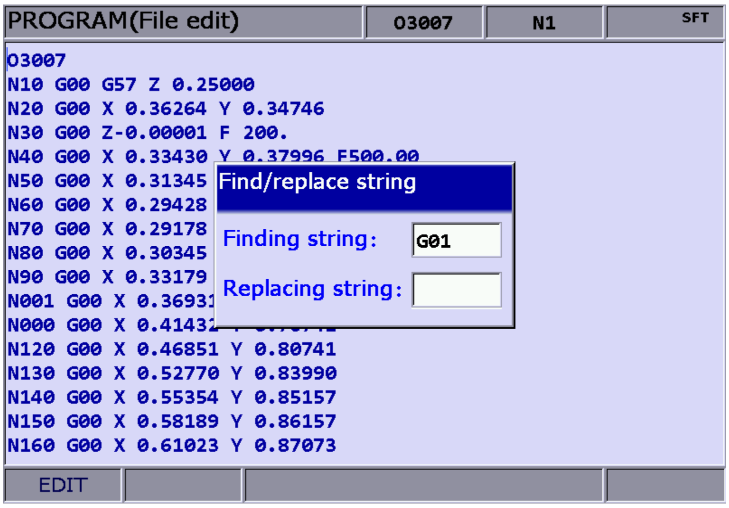


Figure 7.14.1

7. Enter the desired string to be searched and replaced and press the **ENTER** key. The cursor will move to where the string occurred in the file.

8. The string will be highlighted and the 'Forward', 'Backward', ‘Replaced’, and ‘Replaced all’ options are displayed in the function bar.

9. Repeatedly press the **NEXT** to search the next match. Press the **PREV** to search the previous match.

10. Press the **REPLACE** and the system will replace one single string by the one you entered. Or you can press the **REPLACE ALL** to replace all strings that match the searched one.

11. Press the ◀ key to exit the string search page. The function bar resumes displaying options of file editing

12. Please remember to save the editing result (It can save the file by switching modes, pressing **RESET** or opening another file.)

String searching rule:

String searching rule	
Available editing modes	EDIT mode
Size limit of editable files	Only the file with size less than 3MB can be edited.

7.14.3 Block starting/ending point

This function simplifies file editing in case a large section of program modification is required. Users can define a block by defining its starting and ending point with the cursor then edit the block with delete, copy and paste functions.

See the steps below:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. Press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to go to the destination in the second or third layer in the disk.
4. Select the desired G code file. Then, press the **ENTER** key to open the file and enter the screen for file editing.
5. Press the **B start** to set the current cursor position as the starting point of the block.
6. Use ↑, ↓, ←, and → keys to move the cursor to the position as desired block end.
7. Press the **B end** to set the current cursor position as the end of the block. See the figure below.

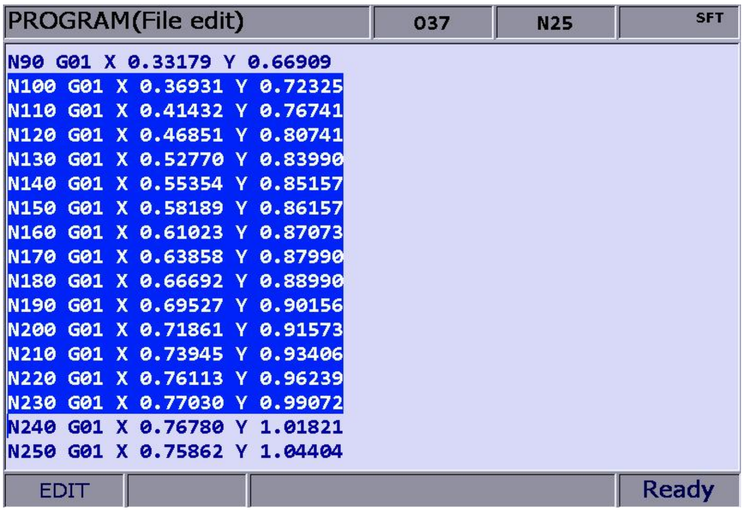


Figure 7.14.2

8. Follow Step 5 ~ 7 and press the **DEL** to delete the text in the given block.
9. Follow Step 5 ~ 7 and press the **Copy** to copy text in this block. Then, move the cursor to the desired area for pasting and press the **Paste** to paste the selected text.

7.14.4 Delete (lines and blocks)

This function deletes the entire line where the cursor is located or the block set up earlier. It also can delete the text of the entire block with the setting of block at starting and ending points.
See the steps described below:

7

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. Press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to go to the destination in the second or third layer in the disk.
4. Select the desired G code file. Then, press the **ENTER** key to open the file and enter the screen for file editing.
5. Move the cursor to the line to be deleted and press the **DEL** to delete the entire line.
6. Delete a program block in the same way as described in Section 7.14.3. See Step 8 for defining the starting and ending points of a block.

7.14.5 Copy and paste (line and block)

Move the cursor to the desired line. Then, press the copy function key and paste the text to the selected location; both copy and paste keys have to be used to fulfill the function. It allows users to copy the text in a single line but also the entire block.
See the operation steps described below:

1. Set the system to **EDIT mode**.
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. Press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to go to the destination in the second or third layer in the disk.
4. Select the desired G code file. Then, press the **ENTER** key to open the file and enter the screen for file editing.
5. Move the cursor to the desired line for copying and press the **Copy**.
6. Move the cursor to the target location and press the **Paste** to paste the line.
7. The entire block can be copied as described in Section 7.14.3. See step 9 for defining the starting and ending points of a block.

7.14.6 Undo

Users can press the undo key to cancel previous editing operations for up to seven steps.

See the steps described below:

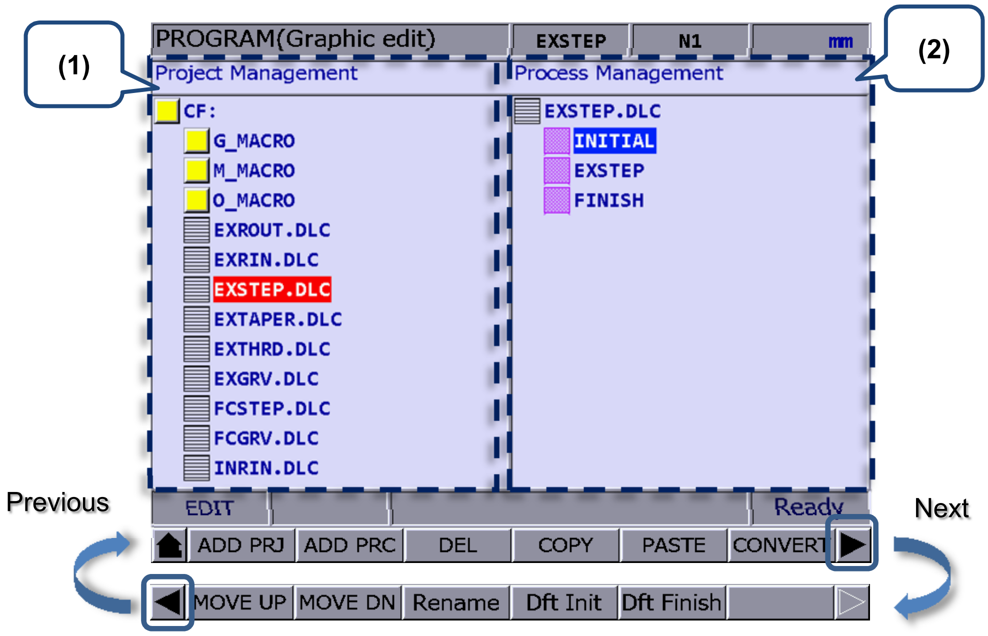
1. Set the system to **Edit mode**.
2. Press the **PRG** key to switch to screen of [PROGRAM].
3. Press ↑, ↓, **PAGE UP**, or **PAGE DN** keys to move the cursor and press the **ENTER** key to go to the destination in the second or third layer in the disk.
4. Select the desired G code file. Then, press the **ENTER** key to open the file and enter the screen for file editing.
5. Press the **Undo** to undo the last action.

7.15 Graphic Edit (Turning System)

7.15.1 Purpose

The graphic programming (Graphic edit) function of the turning system is to provide the graphical HMI for users to choose the machining method and input the relevant machining parameters so as to convert them into the corresponding machining macro programs. It saves the procedure of writing machining programs and calculation as well as the cost for making machining path by CAM.

7.15.2 Project management



(1)Project Management；(2)Process Management

■ **Create a project (ADD PRJ)**

Go to Graphic edit page and click **ADD PRJ**. When the dialogue box pops up, input the file name and press **ENTER** to create a new project. The file extension for graphic programming is **.DLC**. You can manage this project anytime by opening the .DLC project file. With this file, you will be able to find the machining content and modify it or convert it into machining programs.

If you press **ADD PRJ** in GRA edit page and a message “**Please back to project part.**” pops up, press ⬅ to go back to the project management page.

■ **Add new working type**

You have to open a project file before adding a new working type. If no project file is selected, a window showing “**Please open a file!**” will pop up when pressing **ADD PRC**.

Press **ENTER** on the specified project file and you can enter the GRA edit page. Each created project file has two default code, INITIAL and FINISH, which will be regarded as the beginning and end code of the project. Users can add or edit the code between these two default codes. To select another project file, click on to exit the GRA edit page.

- **Delete (DEL)**

You can click on the **DEL** key to delete the specified project file or working type but you cannot delete the default working types, **INITIAL** and **FINISH**. Move the cursor to the project file to be deleted and press **DEL**. Then, press **Y** in the dialogue box to delete the file.

- **Copy and Paste (COPY/PASTE)**

This function copies and pastes project files and working types. Move the cursor to the file to be copied and click on **Copy** and then click **Paste**. Next, press ENTER and copy and paste is done.

- **Convert a file (CONVERT)**

This function is to convert the specified **.DLC** file into **.NC** file. Once the conversion is completed, you will be directed to the program editing page.

Note:

1. If you haven't specified the disc type in the file management page, the message "Please return to the program file management page to select the disc!" will pop up. In this case, you have to go to the previous page to specify the disc to be saved in order to do the conversion.
2. When converting the file, you have to specify a **.DLC** file and click ENTER to open it. Otherwise, the message "**Please open one DLC file！**" will pop out.

- **MOVE UP/ MOVE DN**

This function is available only in the machining management page. Select the project file first and go to its **Process Management** page. Move the cursor to the working type which sorting has to be changed. Click on MOVE UP or MOVE DN and the sorting will be changed.
Please note that this function will not change the sequence of **INITIAL** and **FINISH**.

- **Rename**

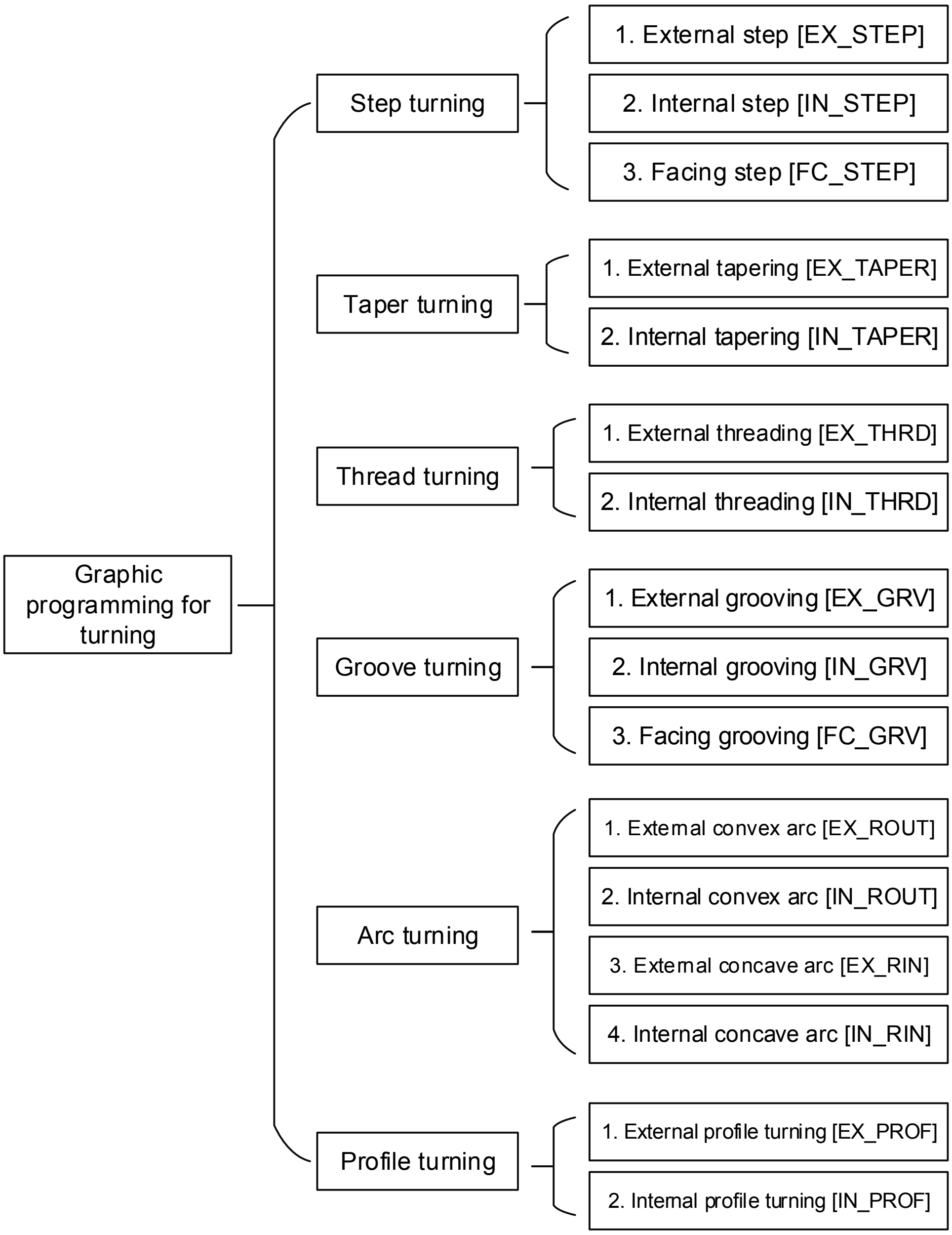
This function key renames the specified project file or working type. However, it cannot rename the default ones, "**INITIAL**" and "**FINISH**". Move the cursor to the project file or working type to be renamed and click on the **Rename** key. Then, input the new file name and press ENTER in the pop-up dialogue box and the rename is completed.

- **Default Initial (Dft Init) & Default Finish (Dft Finish)**

The Default INITIAL function can change the default content of INITIAL. And the Default FINISH function can change the default content of FINISH.The modified content of INITIAL and FINISH will be displayed in the project that is created next time. The old project file remains intact.

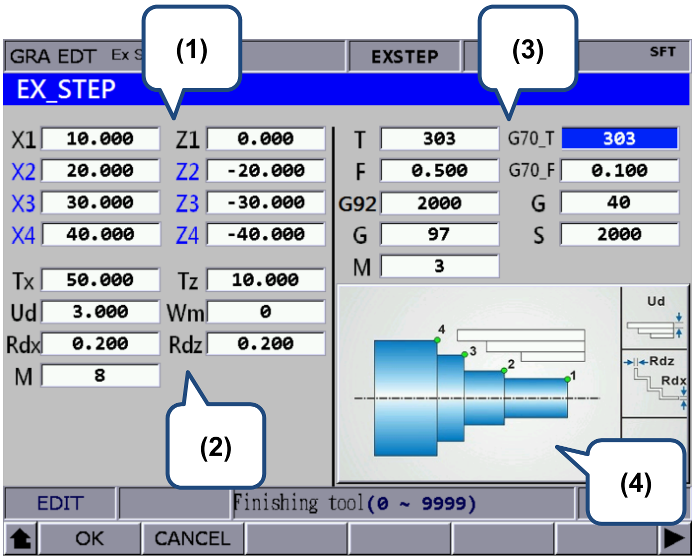
7.15.3 Machining method of graphic programming for turning

Current working types



7

■ **External step [EX_STEP]**



(1) Input the 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 fields are for setting three stages of step turning. When less than 3 stages of step are required, fields that show blue text should be blank. Please press the **BACK SPACE** or **DEL** key to delete and press the **ENTER** key to confirm. You can refer to illustration displayed in section (4) for each step point.

(2) Parameters setting for workpiece required machining

Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Amount of rough cut (Absolute coord. Value of X-axis)	Wm_	Cutting setting (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Rdx_	Reserved amount of finish cut (Absolute coord. Value of X-axis)	Rdz_	Reserved amount of finish cut (Absolute coord. Value of Z-axis)
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

Ud represents the distance of each roughing cut on X-axis.; Rdx represents the reserved distance for finishing cut, which only can be positive value.

Wm is used for selecting the machining mode. If it is set to 0, the system will do peck turning; if Wm is set to 1, the system will execute linear turning all the way to the bottom; if Wm is set to 2, then the system will perform finish cut according to the given amount.

Tx and Tz are used for setting the coordinate of tool-changed position. When the machining process is complete, it will move to the position specified by Tx and Tz for changing the tool of the next cycle.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate

Item	Description	Item	Description
T_	Tool number + Tool compensation number Input format: TXXXX	G70_T	The tool that applied in G70 canned cycle of finish cut
F_	Axial feed rate Unit: mm/min and mm/rev.	G70_F	The feed rate in G70 canned cycle of finish cut
G92_	Limit of the max. spindle speed	G_	Tool nose radius compensation (40 = Disable the compensation / 41 = radius compensation on left / 42 = radius compensation on right)
G_	Spindle speed control mode (G96 = constant surface speed / G97 = constant speed)	S_	Spindle peed
M_	Spindle control (3 = forward / 4 = backward / 5 = stop)	-	-

T field enables users to select the machining tool. Four numbers are in total. You should input the tool number for the first two numbers and tool compensation number for the last two. See the figure above. 303 means cutter No. 3 is applied and enable the compensation of tool No. 3.

G70_T represents the applied tool number when executing G70 canned cycle of finish cut. Its input method is identical to field T.

G70_F is the applied cutter feed rate when executing G70 canned cycle of finish cut.

G_ is used for tool nose radius compensation. G40 means to disable the compensation function; G41 means to enable the compensation on left; And G42 means to enable the compensation on right. Before applying this function, you have to input the position of tool nose and the value of tool nose radius.

G96 and G97 are used for setting spindle speed control mode. Input G96 in this field will enable the function of constant surface speed. Field S is speed setting. Its unit is m/min or feet/min; Input G97 will enable the function of constant speed. Field S is the speed per minute and its unit is rpm.

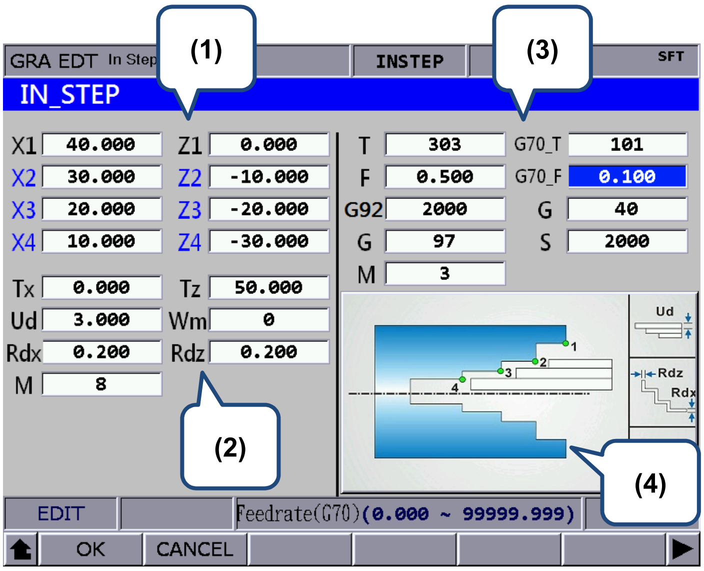
G92 is the max. speed limit of the spindle. If the speed command exceeds the setting value,

7

it will run with the max. speed set by G92.

(4) Illustration

■ **Internal step [IN_STEP]**



(1) Input the coordinates of each step point (Fields with blue texts can be left blank and it means the point does not exist.)

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

You can refer to illustration displayed in section (4) for each point's position.

(2) Parameters setting for workpiece required machining

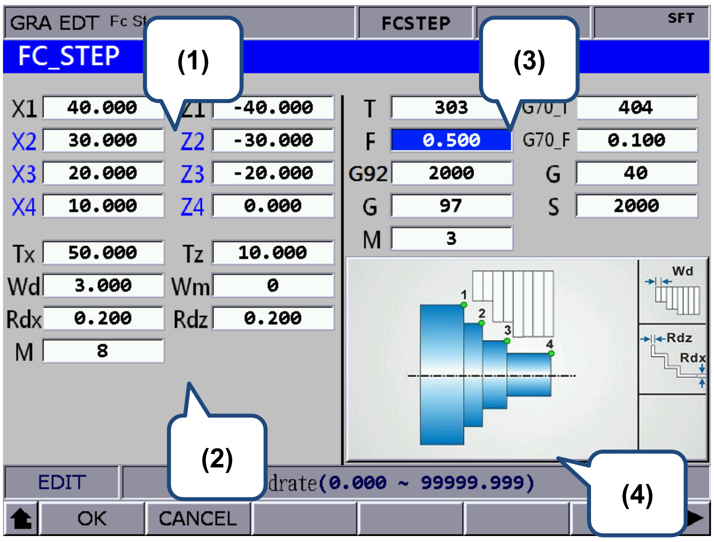
Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Amount of rough cut (Absolute coord. Value of X-axis)	Wm_	Cutting setting (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Rdx_	Reserved amount of finish cut (Absolute coord. Value of X-axis)	Rdz_	Reserved amount of finish cut (Absolute coord. Value of Z-axis)
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate

Item	Description	Item	Description
T_	Tool number + Tool compensation number Input format: TXXXX	G70_T	The tool that applied in G70 canned cycle of finish cut
F_	Axial feed rate Unit: mm/min and mm/rev.	G70_F	The feed rate in G70 canned cycle of finish cut
G92_	Limit of the max. spindle speed	G_	Tool nose radius compensation (40 = Disable the compensation / 41 = radius compensation on left / 42 = radius compensation on right)
G_	Spindle speed control mode (G96 = constant surface speed / G97 = constant speed)	S_	Spindle peed
M_	Spindle control (3 = forward / 4 = backward / 5 = stop)	-	-

(4) Illustration

■ **Facing step [FC_STEP]**



(1) Input the coordinates of each step point (Fields with blue texts can be left blank and it means the point does not exist.)

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

You can refer to illustration displayed in section (4) for each point's position.

(2) Parameters setting for workpiece required machining

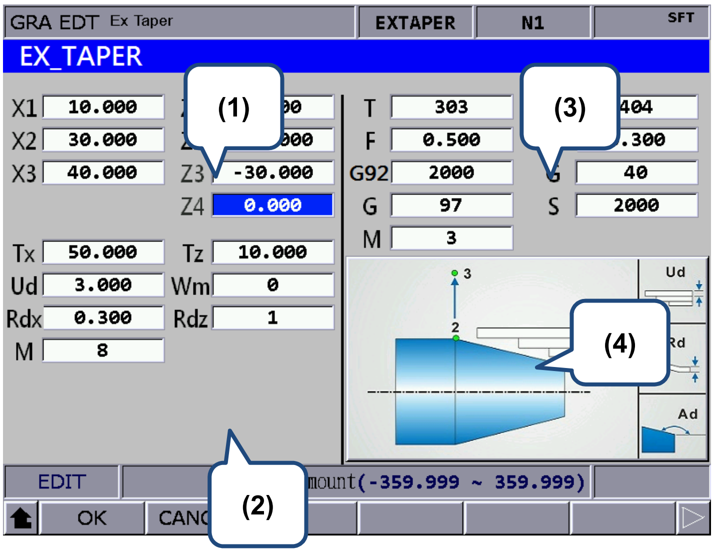
Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Amount of rough cut (Absolute coord. Value of X-axis)	Wm	Cutting setting (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Rdx_	Reserved amount of finish cut (Absolute coord. Value of X-axis)	Rdz_	Reserved amount of finish cut (Absolute coord. Value of Z-axis)
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate

Item	Description	Item	Description
T_	Tool number + Tool compensation number Input format: TXXXX	G70_T	The tool that applied in G70 canned cycle of finish cut
F_	Axial feed rate Unit: mm/min and mm/rev.	G70_F	The feed rate in G70 canned cycle of finish cut
G92_	Limit of the max. spindle speed	G_	Tool nose radius compensation (40 = Disable the compensation / 41 = radius compensation on left / 42 = radius compensation on right)
G_	Spindle speed control mode (G96 = constant surface speed / G97 = constant speed)	S_	Spindle speed
M_	Spindle control (3 = forward / 4 = backward / 5 = stop)	-	-

(4) Illustration

■ External tapering [EX_TAPER]



(1) Input the coordinates of each step point (Fields with blue texts can be left blank and it means the point does not exist.)

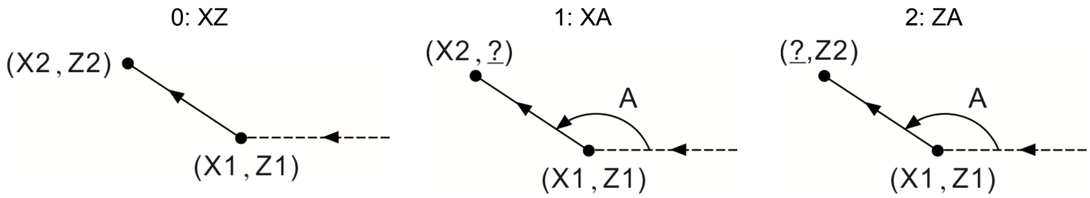
Item	Description	Item	Description
1st point	X1_ , Z1_	2nd point	X2_ , Z2_
3rd point	X3_ , Z3_	Ad	Tapered angle of inclination

You can refer to illustration displayed in section (4) for each point's position.

(2) Parameters setting for workpiece required machining

Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Amount of rough cut (Absolute coord. Value of X-axis)	Wm	Cutting setting (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Rd_	Reserved amount of finish cut (Absolute coord. Value of X-axis)	Kd_	Taper format (0 = XZ, 1 = XA, 2 = ZA)
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

Kd: Taper format

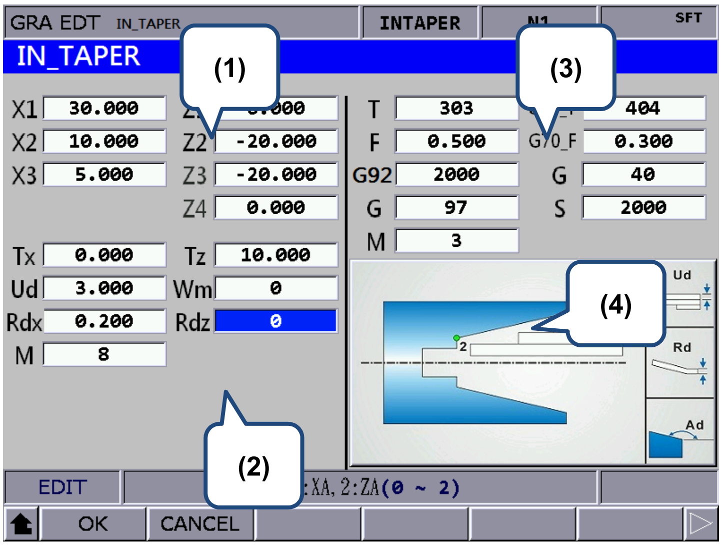


(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

7

(4) Illustration

■ **Internal tapering [IN_TAPER]**



(1) Input the coordinates of each step point (Fields with blue texts can be left blank and it means the point does not exist.)

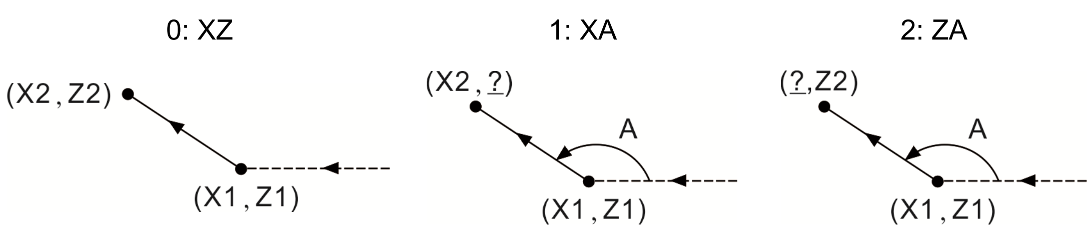
Item	Description	Item	Description
1st point	X1_ , Z1_	2nd point	X2_ , Z2_
3rd point	X3_ , Z3_	Ad	Tapered angle of inclination

You can refer to illustration displayed in section (4) for each point's position.

(2) Parameters setting for workpiece required machining

Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Amount of rough cut (Absolute coord. value of X-axis)	Wm	Machining mode selection (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Rd_	Reserved amount of finish cut	Kd_	Taper format (0 = XZ, 1 = XA, 2 = ZA)
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

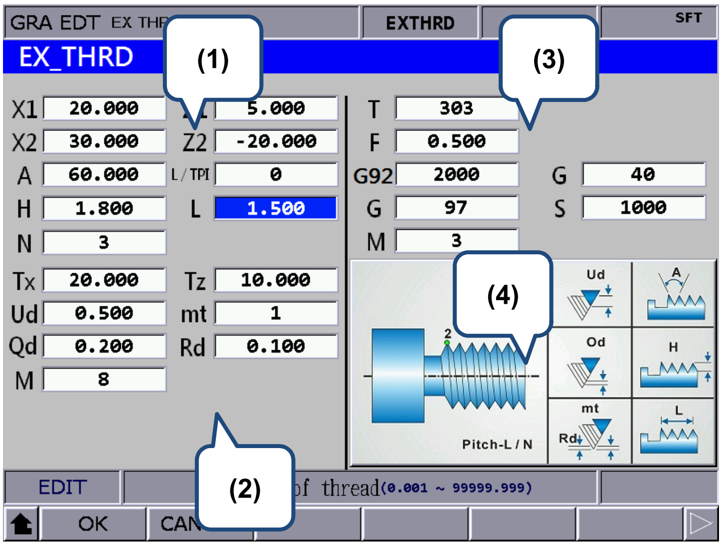
Kd: Taper format



(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

■ **External threading [EX_THRD]**



(1) Input the coordinates of each step point

Item	Description	Item	Description
1st point	X1_ , Z1_	2nd point	X2_ , Z2_
A	Thread angle	L/TPI	0 = length of thread lead; 1 = thread teeth number of every inch
H	Total thread cutting depth	L	Unit of L/TPI; L = length of thread lead, unit is mm/pitch; TPI = thread teeth number of every inch, unit is teeth/inch
N	Thread screw number		

(2) Parameters setting for workpiece required machining

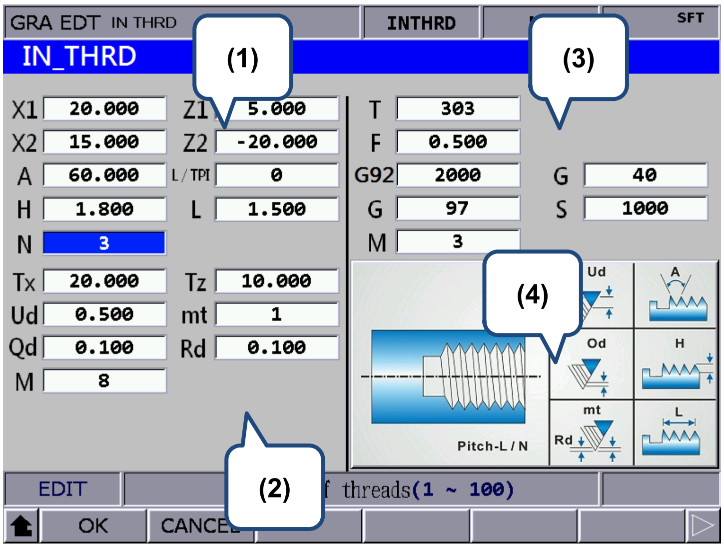
Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	First cutting depth	mt	Number of finish cut
Qd_	Min. cutting depth	Rd_	Reserved amount of finish cut
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

Ud is the first thread cutting depth. When it cuts for N times (N ≧ 2), the system will automatically calculate the cutting depth (Calculation formula: $\mathrm{Ud}\left[\sqrt{n}-\sqrt{(n-1)}\right]$). Qd represents the min. cutting depth. When the cutting depth is smaller than Qd after N times of calculation, the system will regard the setting value of Qd as the feeding amount. Rd is the reserved amount of finish cut and mt is the number of finish cut. Thus, the amount of each finish cut is Rd/mt.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

■ **Internal threading [IN_THRD]**



(1) Input the coordinates of each step point

Item	Description	Item	Description
1st point	X1_ , Z1_	2nd point	X2_ , Z2_
H	Total thread cutting depth	L	Unit of L/TPI; L= length of thread lead, unit is mm/pitch; TPI = thread teeth number of every inch, unit is teeth/inch
N	Thread screw number		

(2) Parameters setting for workpiece required machining

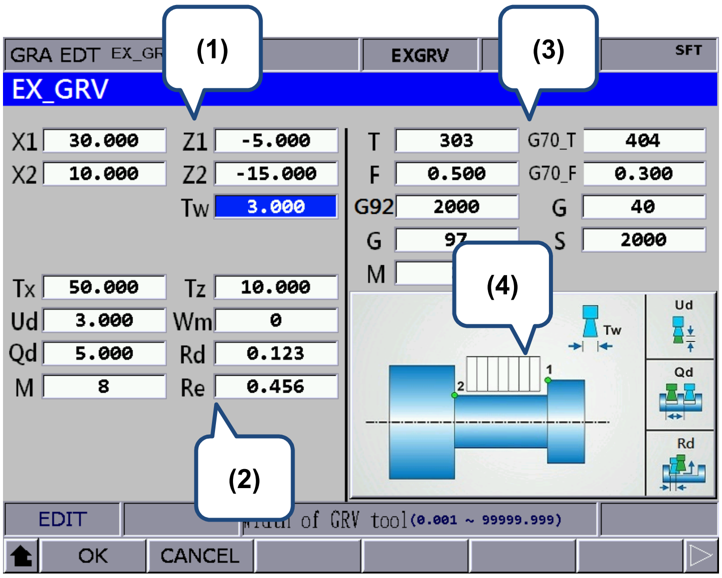
Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	First cutting depth	mt	Number of finish cut
Qd_	Min. cutting depth	Rd_	Reserved amount of finish cut
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

Ud is the first thread cutting depth. When it cuts for N times (n ≧ 2), the system will automatically calculate the cutting depth (Calculation formula: $\mathrm{Ud}\left[\sqrt{n}-\sqrt{(n-1)}\right]$). Qd represents the min. cutting depth. When the cutting depth is smaller than Qd after N times of calculation, the system will regard the setting value of Qd as the feeding amount. Rd is the reserved amount of finish cut and mt is the number of finish cut. Thus, the amount of each finish cut is Rd/mt.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

■ **External Grooving [EX_GRV]**



(1) Input the coordinates of each step point

Item	Description	Item	Description
1st point	X1_ , Z1_	2nd point	X2_ , Z2_
Tw	Width of grooving tool	-	-

(2) Parameters setting for workpiece required machining

Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Peck amount per time of axis X	Wm	Machining mode selection (0 = peck turning / 1 = straight turning / 2 = finish turning)
Qd_	Amount of feed per time (Z-axis)	Rd_	The setback amount on Z-axis after

Item	Description	Item	Description
			axis X completes cutting.
M_	Switch of cutting fluid (8 = On / 9 = Off)	Re_	The retraction amount on X-axis after each peck

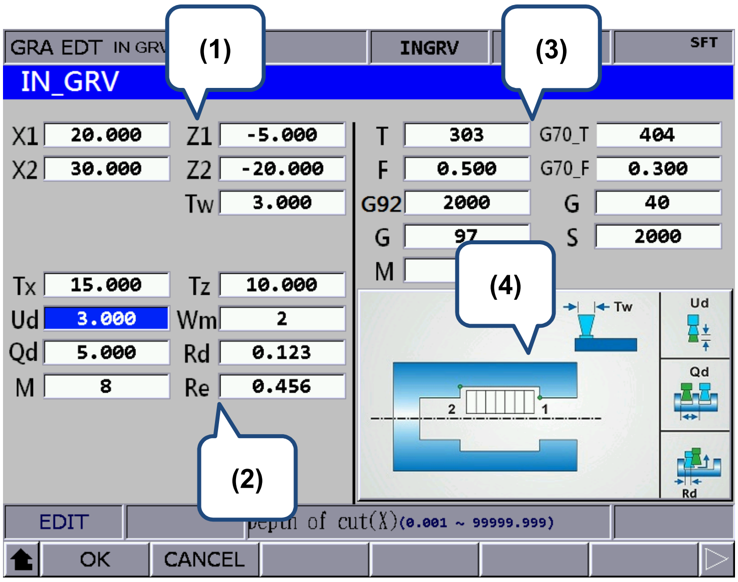
Ud is used for setting the feeding amount of each peck cutting on X-axis. Qd is the amount of feed per time in Z-axis direction. Cd is used for setting the setback amount on Z-axis when axis X completes cutting. The input value must be greater than 0.

Wm is used for selecting the groove turning mode. If it is set to 0, the system will perform peck cutting; if it is set to 1, the system will straightly cut down to the bottom; if the value is 2, the system will perform finish cut according to the given amount.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

■ **Internal grooving [IN_GRV]**



(1) Input the coordinates of each step point

Item	Description	Item	Description
1st point	X1_ , Z1_	2nd point	X2_ , Z2_
Tw	Width of grooving tool	-	-

(2) Parameters setting for workpiece required machining

Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Peck turning amount per time of	Wm	Machining mode selection

Item	Description	Item	Description
	axis X		(0 = peck turning / 1 = straight turning / 2 = finish turning)
Qd_	Amount of feed per time (Z-axis)	Rd_	The setback amount on Z-axis after axis X completes cutting.
M_	Switch of cutting fluid (8 = On / 9 = Off)	Re_	The retraction amount of X-axis after each peck

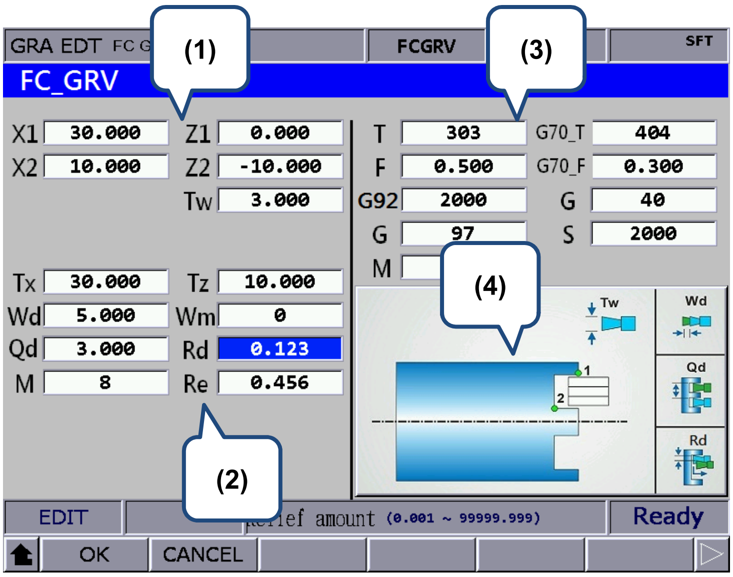
Ud is used for setting the feeding amount of each peck cutting on X-axis. Qd is the feeding amount per time in Z-axis direction. Cd is used for setting the setback amount of Z-axis when X-axis completes cutting. The input value must be greater than 0.

Wm is used for selecting the groove turning mode. If it is set to 0, the system will perform peck cutting; if it is set to 1, the system will straightly cut down to the bottom; if the value is 2, the system will perform finish cut according to the given amount.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

- **Face grooving [FC_GRV]**



(1) Input the coordinates of each step point

Item	Description	Item	Description
1st point	X1_ , Z1_	2nd point	X2_ , Z2_
Tw	Width of grooving tool	-	-

(2) Parameters setting for workpiece required machining

Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Wd_	Peck turning amount per time of axis Z	Wm	Machining mode selection (0 = peck turning / 1 = straight turning / 2 = finish turning)
Qd_	Feeding amount per time (Z-axis)	Rd_	The setback amount on X-axis after axis Z completes cutting.
M_	Switch of cutting fluid (8 = On / 9 = Off)	Re_	The retraction amount on Z-axis after each peck

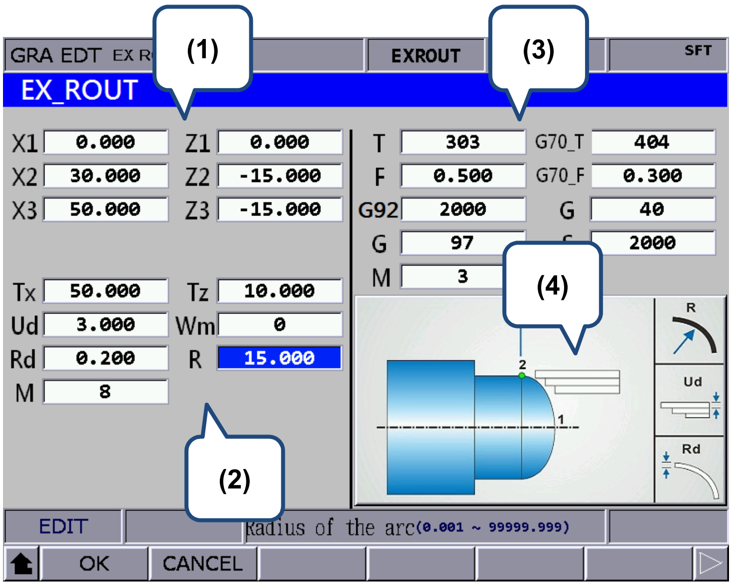
Wd is used for setting the feeding amount of each peck cutting on Z-axis. Qd is the feeding amount per time in X-axis direction. Cd is used for setting the setback amount on X-axis when axis Z completes cutting. The input value must be more than 0.

Wm is used for selecting the groove turning mode. If it is set to 0, the system will perform peck cutting; if it is set to 1, the system will straightly cut down to the bottom; if the value is 2, the system will perform finish cut according to the given amount.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

- **External convex arc [EX_ROUT]**



(1) Input the coordinates of each step point

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

(2) Parameters setting for workpiece required machining

Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Amount of rough cut in X-axis direction	Wm	Machining mode selection (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Rd_	Reserved amount of finish cut	R_	Radius of circular arc
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

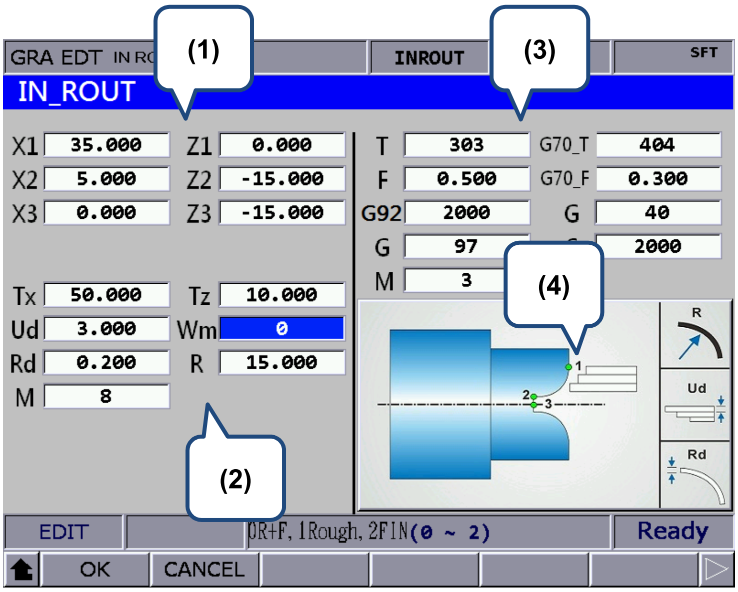
R is the radius of circular arc. Make sure the coordinates of X1, Z1, X2 and Z2 are all correct when inputting value R. If the input value is inappropriate, an alarm will occur when machining operation is enabled. Ud represents the depth of each cut in X-axis direction. Rd is used for setting the reserved amount of finish cut.

Wm is used for selecting the machining mode. When Wm is set to 0, the system will complete rough and finish cut for one time. If Wm is set to 1, the system only performs rough cut and saves the reserved amount of finish cut. If Wm is set to 2, it only performs finish cut according to the given amount.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

■ **Internal convex arc [IN_ROUT]**



(1) Input the coordinates of each step point

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

(2) Parameters setting for workpiece required machining

Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Amount of rough cut in X-axis direction	Wm	Machining mode selection (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Rd_	Reserved amount of finish cut	R_	Radius of circular arc
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

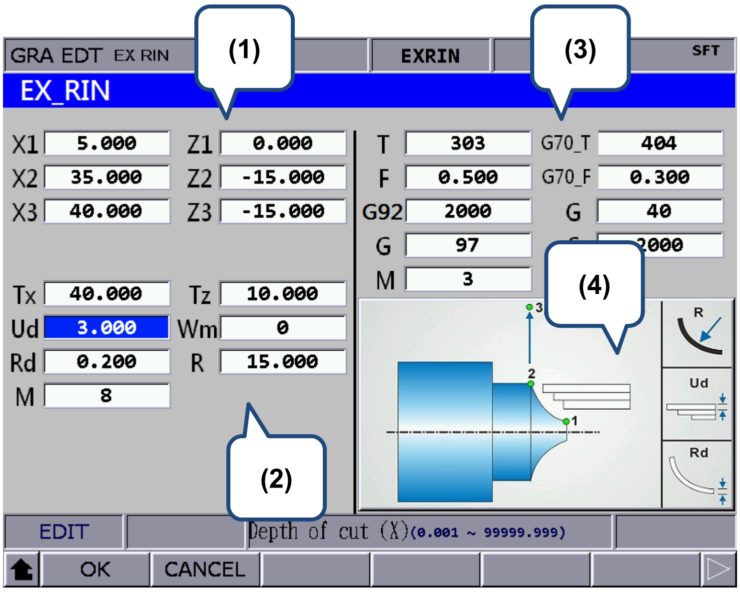
R is the radius of circular arc. Make sure the coordinates of X1, Z1, X2 and Z2 are all correct when inputting value R. If the input value is inappropriate, an alarm will occur when machining operation enabled. Ud represents the depth of each cut in X-axis direction. Rd is used for setting the reserved amount of finish cut.

Wm is used for selecting the machining mode. When Wm is set to 0, the system will complete rough and finish cut for one time. If Wm is set to 1, the system only performs rough cut and saves the reserved amount of finish cut. If Wm is set to 2, it only performs finish cut according to the given amount.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

■ **External concave arc [EX_RIN]**



(1) Input the coordinates of each step point

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

(2) Parameters setting for workpiece required machining

Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Amount of rough cut in X-axis direction	Wm	Machining mode selection (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Rd_	Reserved amount of finish cut	R_	Radius of circular arc
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

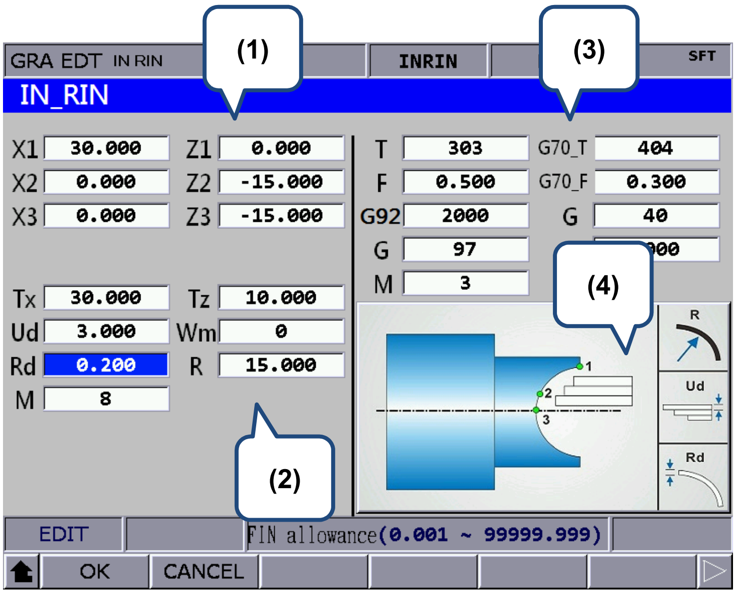
R is the radius of circular arc. Make sure the coordinates of X1, Z1, X2 and Z2 are all correct when inputting value R. If the input value is inappropriate, an alarm will occur when machining operation is enabled. Ud represents the depth of each cut in X-axis direction. Rd is used for setting the reserved amount of finish cut.

Wm is used for selecting the machining mode. When Wm is set to 0, the system will complete rough and finish cut for one time. If Wm is set to 1, the system only performs rough cut and saves the reserved amount of finish cut. If Wm is set to 2, it only performs finish cut according to the given amount.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

■ **Internal concave arc [IN_RIN]**



(1) Input the coordinates of each step point

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

(2) Parameters setting for workpiece required machining

Item	Description	Item	Description
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis

Item	Description	Item	Description
Ud_	Amount of rough cut in X-axis direction	Wm	Machining mode selection (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Rd_	Reserved amount of finish cut	R_	Radius of circular arc
M_	Switch of cutting fluid (8 = On / 9 = Off)	-	-

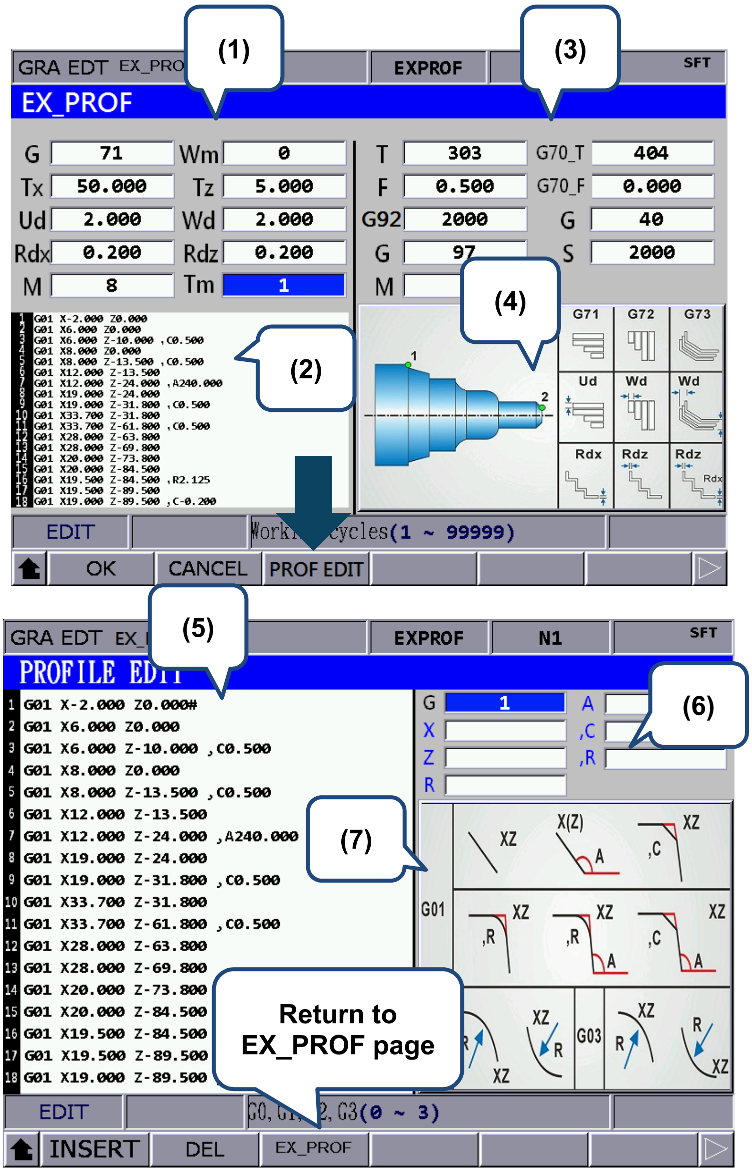
R is the radius of circular arc. Make sure the coordinates of X1, Z1, X2 and Z2 are all correct when inputting the value R. If the input value is inappropriate, an alarm will occur when machining operation is enabled. Ud represents the depth of each cut in X-axis direction. Rd is used for setting the reserved amount of finish cut.

Wm is used for selecting the machining mode. When Wm is set to 0, the system will complete rough and finish cut for one time. If Wm is set to 1, the system only performs rough cut and saves the reserved amount of finish cut. If Wm is set to 2, it only performs finish cut according to the given amount.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

■ **External profile turning [EX_PROF]**



(1) Machining parameters setting

Item	Description	Item	Description
G_	Rough turning cycle	Wm	Machining mode selection (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Amount of rough cut (X-axis)	Wd_	Amount of rough cut (Z-axis)
Rdx_	Reserved amount of finish cut (X-axis)	Rdz_	Reserved amount of finish cut (Z-axis)
M_	Switch of cutting fluid (8 = On / 9 = Off)	Tm_	Number of turning cycle

G71/G72/G73 profile turning mode selection:

G71 is used for turning longer and thinner bar in Z-axis direction; G72 is applicable to shorter and thicker bar turning; And G73 is used for turning the forging or blank machining workpiece.

Input the value in Ud field when applying G71 and G73 modes. G71 is used for setting the depth of each rough cut in X-axis direction. G73 is used for setting the total cut amount in X-axis direction.

Use Wd field when applying G72 and G73. G72 is used for setting the depth of each rough cut in Z-axis direction. G73 is used for setting the total cut amount in Z-axis direction.

Tm field is for G73 only. It represents the cutting cycle number. Divide the cutting amount set by Ud and Wd by Tm, the result will be the depth of each cut.

(2) **Profile Edit** window: This section displays the code generated in Profile Edit window. Users can directly enter G code or use ⇧ and ⇩ keys to move up or down for editing.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

(5) **Profile Edit** window: Its function is identical to section (2).

(6) Program editing: Input the parameter of each G code. Then, press **INSERT** to insert the program.

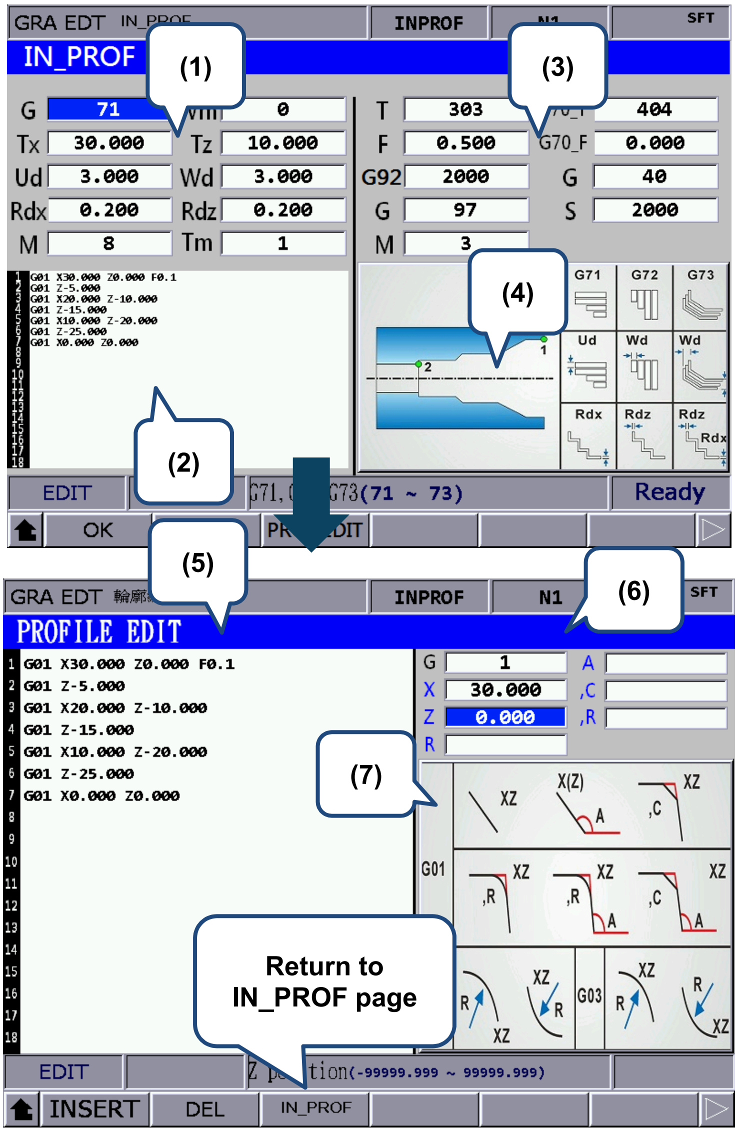
(7) Illustration of each instruction: When editing the machining path, users can create the path according to the instruction format mentioned in the above table. This section has three types, G01, G02 and G03.

G01 type includes: 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_.

G02 type is G02X_Z_R_ and G03 type is G03X_Z_R_.

C means auto chamfer and R means auto rounding off.

■ **Internal profile turning [IN_PROF]**



(1) Machining parameters setting

Item	Description	Item	Description
G_	Rough turning cycle	Wm	Machining mode selection (0 = roughing + finishing / 1 = roughing / 2 = finishing)
Tx_	Tool-changed position on X-axis	Tz_	Tool-changed position on Z-axis
Ud_	Amount of rough cut (X-axis)	Wd_	Amount of rough cut (Z-axis)
Rdx_	Reserved amount of finish cut (X-axis)	Rdz_	Reserved amount of finish cut (Z-axis)
M_	Switch of cutting fluid (8 = On / 9 = Off)	Tm_	Number of turning cycle

G71/G72/G73 profile turning mode selection:

G71 is used for turning longer and thinner bar in Z-axis direction; G72 is applicable to shorter and thicker bar turning; And G73 is used for turning the forging or blank machining workpiece.

Input the value in Ud field when applying G71 and G73 modes. G71 is used for setting the depth of each rough cut in X-axis direction. G73 is used for setting the total cut amount in X-axis direction.

Use Wd field when applying G72 and G73. G72 is used for setting the depth of each rough cut in Z-axis direction. G73 is used for setting the total cut amount in Z-axis direction.

Tm field is for G73 only. It represents the cutting cycle number. Divide the cutting amount set by Ud and Wd by Tm, the result will be the depth of each cut.

(2) **Profile Edit** window: This section displays the code generated in Profile Edit window. Users can directly enter G code or use ⬆ and ⬇ keys to move up or down for editing.

(3) Parameters setting for tool compensation, tool nose radius compensation, speed and feed rate. Please follow the setting mentioned in EX_STEP section (3).

(4) Illustration

(5) **Profile Edit** window: Its function is identical to section (2).

(6) Program editing: Input the parameter of each G code. Then, press **INSERT** to insert the program.

(7) Illustration of each instruction: When editing the machining path, users can create the path according to the instruction format mentioned in the above table. This section has three types, G01, G02 and G03.

G01 type includes: 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_.

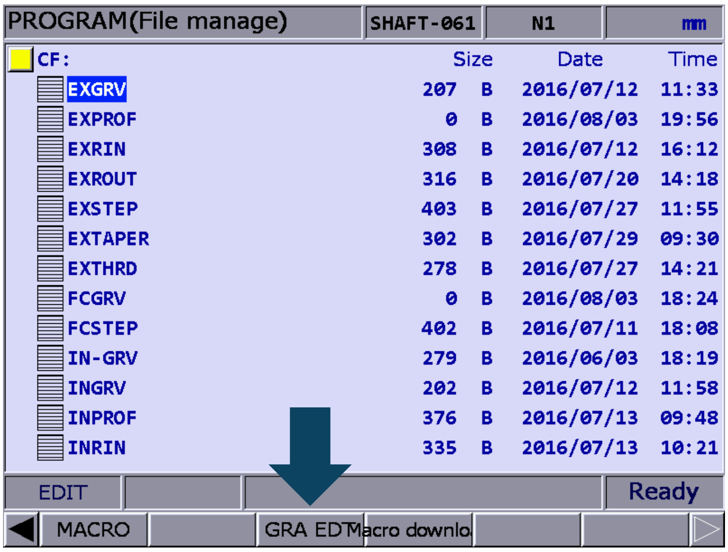
G02 type is G02X_Z_R_ and G03 type is G03X_Z_R_.

C means auto chamfer and R means auto rounding off.

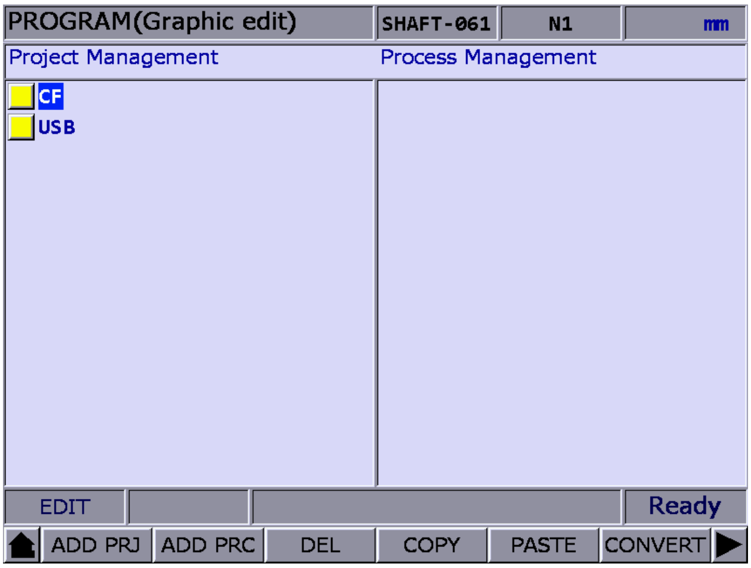
7

7.15.4 Operation steps of graphic programming

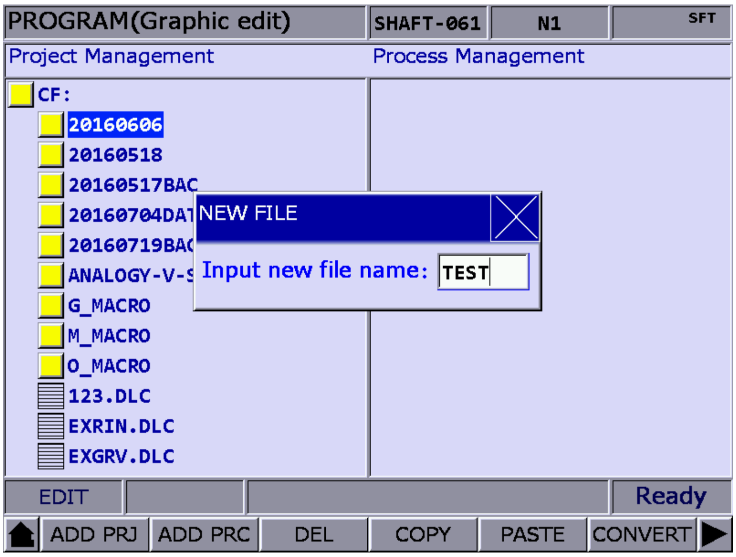
1. Before entering graphic programming page, the NC controller has to be in [EDIT] mode.
2. Find **GRA EDT** (Graphic programming) is in the last page of file manager.



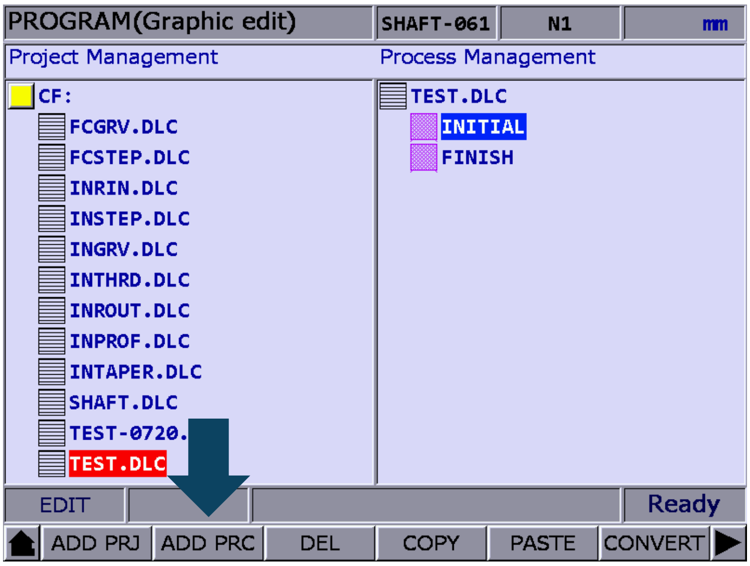
3. Go to GRA EDT page and select the disc type (CF/USB).



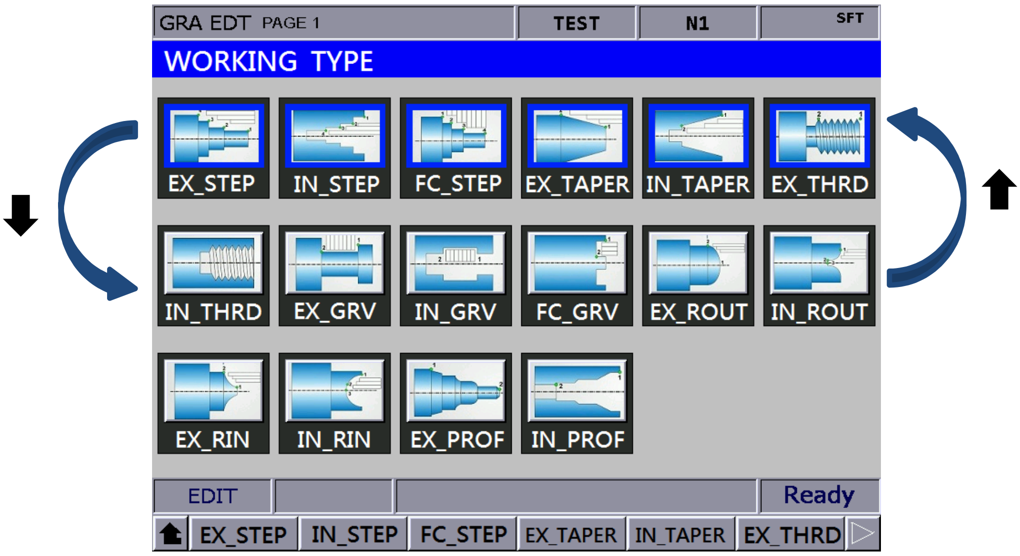
4. Then, select **ADD PRJ** and enter the filename. Press **ENTER** to generate the project file.



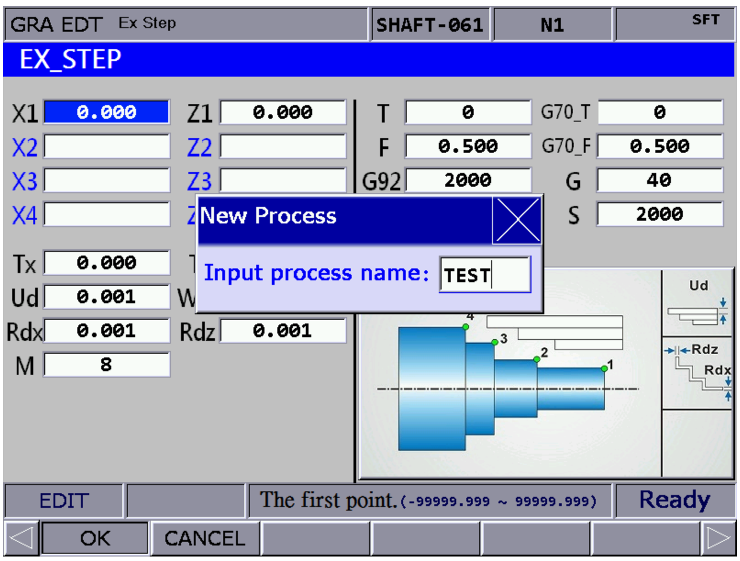
5. Move your cursor to one of the projects. And press **ENTER**.
6. Press **ADD PRC** to enter **GRA EDT** page and select the working type.



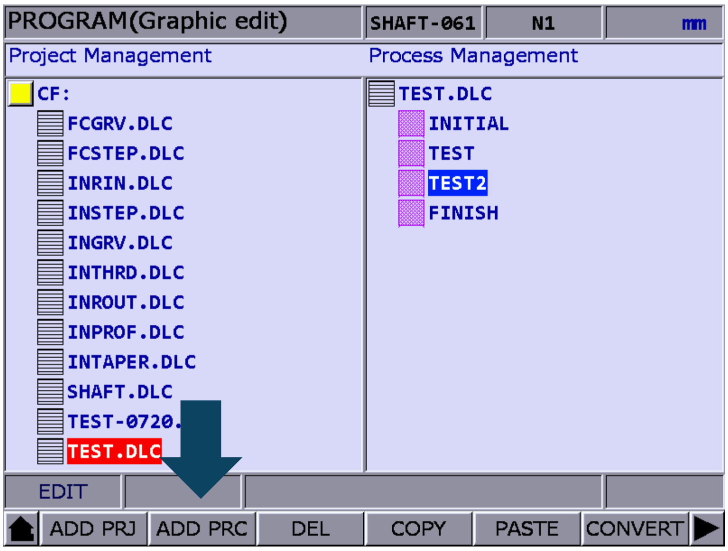
7. The system provides 16 working types. 6 types are in one row, which corresponds to functions keys respectively. Users can use and keys to switch the row. When the row is switched, the editing page corresponded to functions keys will be changed.



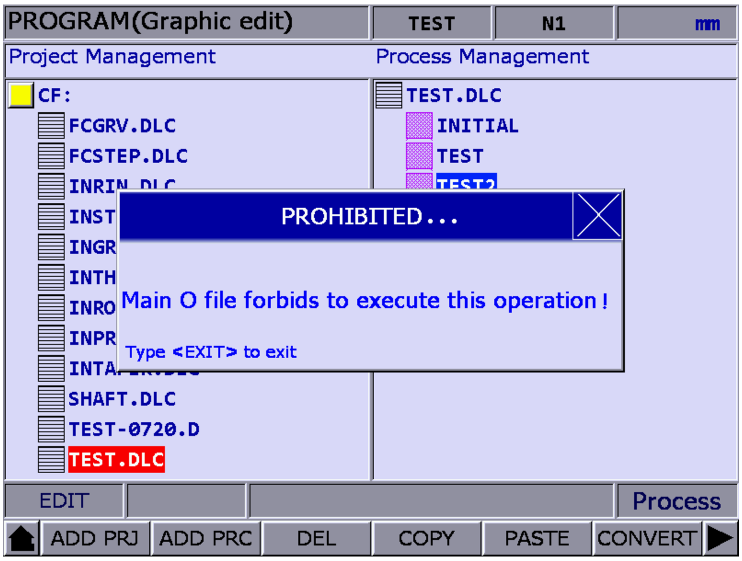
8. Select the working type and press the cooresponding function key to enter its editing page.
9. When completeing the parameters setting, press **OK**. Then, input the process name and press **ENTER** to complete editing.



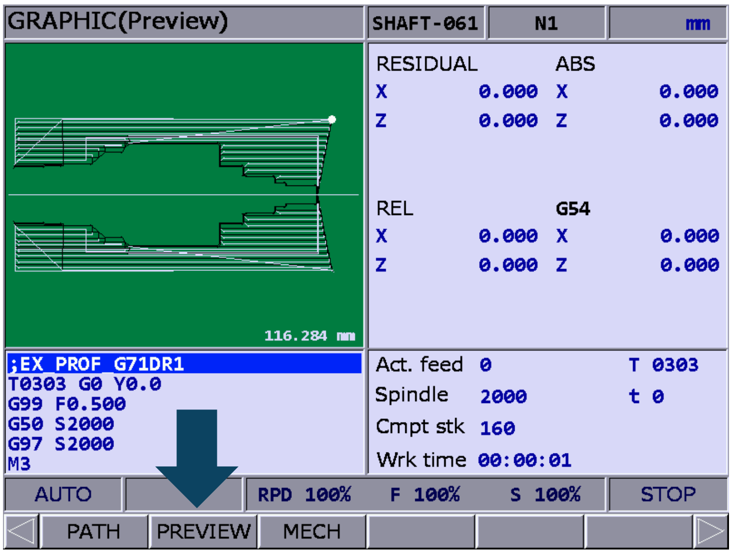
10. If you want to continue with other working types, press **ADD PRC** to insert the requied ones.



11. If all working types editing are complete, press **CONVERT** to convert the file into G code format. Please note that the file cannot be opened as the main file during file conversion. Otherwise, a warning message of “Main O file forbids to execute this operation!“ will pop up.

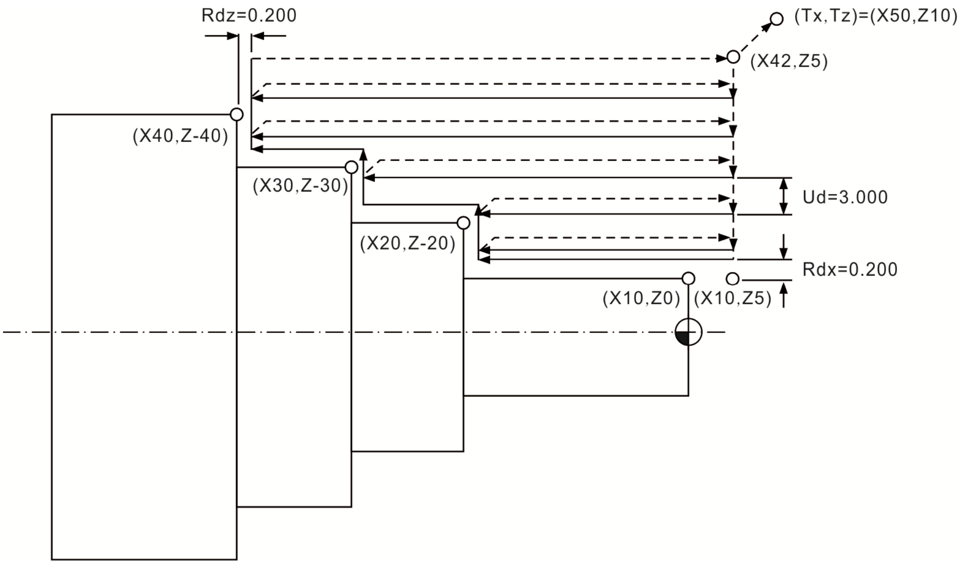


12. After converting the file, you will be directed to **FILE EDIT** page. The system will check the accuracy of the program. Then, apply **PREVIEW** function in GRA page to see if the graphic is applicable for machining. Now, you can perform MPG simulation or machining.



[Example of file conversion: External step]

This example displays the G code file generated by parameters setting in external step.



O0000 // Contents in **INITIAL** will be added to the beginning of the program automatically

;EXstep_DR0

T0303 // Tool number + Tool compensation number

G95 F0.500 // 0.5 mm/feed of every revolution

G92 S2000 // Max. speed limit of the spindle

G97 S1000 // Constant spindle speed

M3 // Spindle runs in forward direction

M8 // Switch of cutting fluid is turned On

G00 X42.000 Z5.000

G42 //Enable tool nose radius compensation

```
G71 U3.000 R0.5    // Amount of rough cut (X-axis)
G71 P210 Q250 U0.200 W0.200    // Reserved amount of finish cut (X- and Z-axis)
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    // The applied tool when performing finish cut
G70 P210 Q250 F0.100 // Finish cut
G00 Z5.000
G40
G00 X50.000 Z10.000    //Tool-changed position
M09
M05
M30    //    Contents in FINISH will be added to the end of the program automatically
```

7.16 Other modes

Auto mode (Auto):

After entering the PRG group screen, the contents of the currently opened G code file will be displayed. Users will be able to view the status information of the currently opened/executed file as well as the line being executed. The PRG group function in auto mode displays information relevant to program execution and coordinates of movements during program running.

See the operation steps below:

1. Press the **PRG** key in **Auto mode** to display status of program running in full screen as illustrated in the figure below.

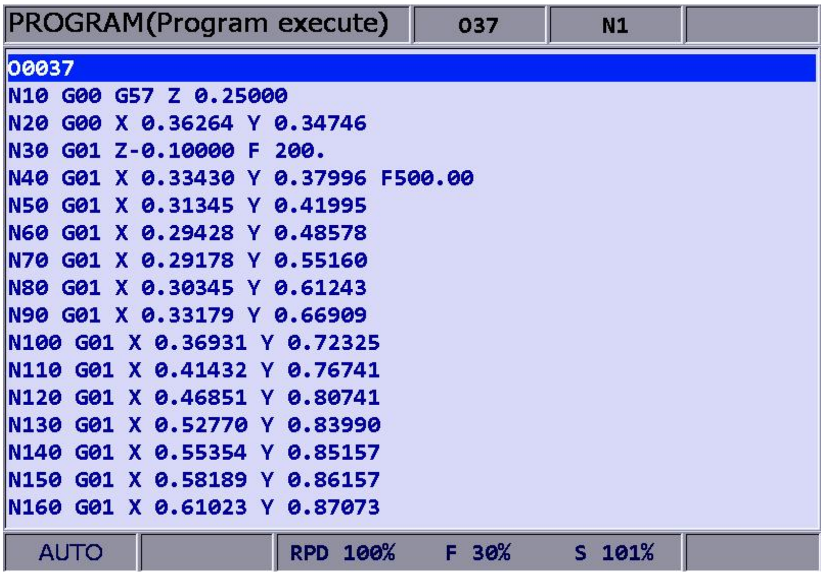


Figure 7.16.1

2. Press the **PRG** key again to switch to the screen which displays both program and coordinates. See the figure below.

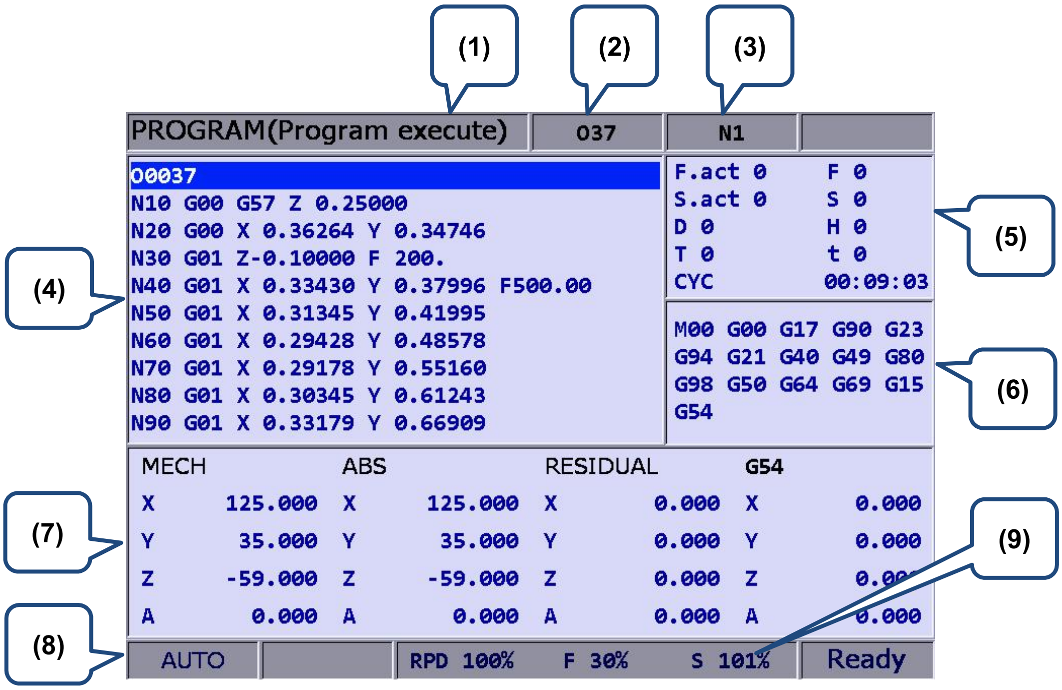


Figure 7.16.2

(1) Current group
(2) Current program
(3) The line being executed
(4) Content of program being executed
(5) F.act: actual feed rate
S.act: actual spindle speed
D: tool radius compensation ID
H: tool length compensation ID
T: tool ID
F: Feed rate
S: Spindle speed
t: pause time
CYC: Single machining time
(6) Current command status
(7) Information of each coordinate system
(8) Current mode
(9) Current motion factors

When program is stopped, the **breakpoint search function** (Search) in auto mode automatically records the line number where it is stopped; it will quickly do the computing and execute the program content prior to the breakpoint. See the figure below for illustration.

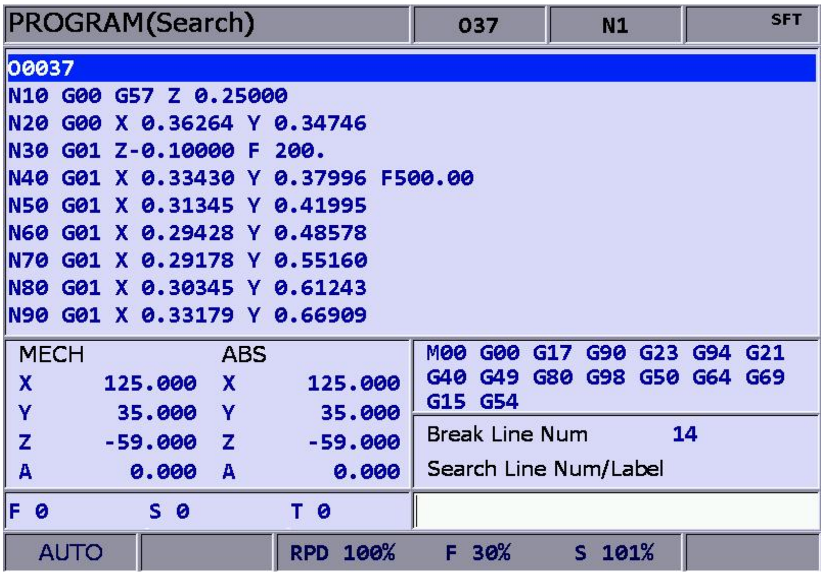


Figure 7.16.3

See the operation steps below:

1. In **AUTO Mode**, press the **PRG** key to enter the screen of [PROGRAM].
2. Press the **START** and the screen for breakpoint search will pop up.
3. Refer to the breakpoint line number shown in the screen, enter the desired program line or sequence number. Then, press the **ENTER** key to complete the setting.
4. Press the **RUN** to quickly execute the program and go to the re-starting line or sequence number.
5. Before executing to the specified line, the controller will automatically execute the program and record its status. The system stops at the breakpoint line and waits for its execution.
6. Press the **Cycle start** key to resume normal program execution.

Note:

1. The system stops program execution when reaching the block after the breakpoint. This block remains unexecuted until the **Cycle start** key is pressed and the system resumes normal operation.
2. Valid search formats are the line number and N number of the program.
3. During program running or breakpoint search function is performing, any request for breakpoint search will be ignored as the system regards it as in program running status.

The **SF setup** function can be used to change the feeding speed (F command) and spindle speed (S command) during G code execution as shown in Figure 7.16.4 below. By using SF setup function and entering the new command value, the speed command is changed when program running.

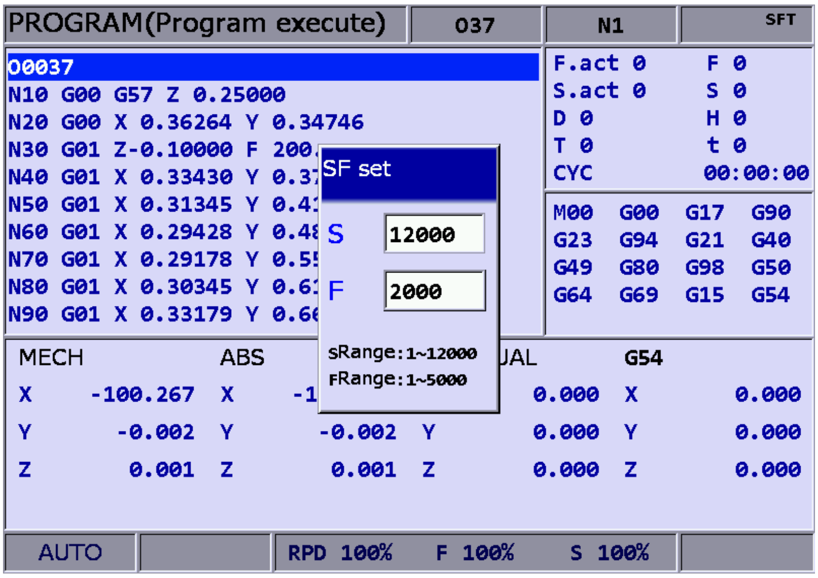


Figure 7.16.4

See the operation steps below:

1. In **AUTO Mode**, press the **PRG** key to enter the screen of [PROGRAM].
2. Press the **SF set** and the dialog box for entering SF command will pop up.
3. Enter the new S or F values then press the **ENTER** key and the speed is changed.

Note:

1. The SF settings are valid during single execution only when the S and F values in the G code remain intact. For a G code that requires repeated execution, it is recommended to edit the program and ensure the accuracy of speed commands in Edit mode.
2. After the S value is set, the spindle speed will be immediately changed in the G code. On the other hand, when the F value is set, the new feed speed (F command) takes effect only after new data in the system buffer is processed.
3. Do not use this function to change the existing speed command for a G code program without applying S and F commands.
4. In SF setting, F command is enabled by the function of “Enable feed rate speed setting “ in parameter No.10017.

By using the bar-code reader, the File Scan function can quickly load in and sequence the machining files that are named by bar code. This saves the time for file searching. The bar-code reader can be installed via the USB port.

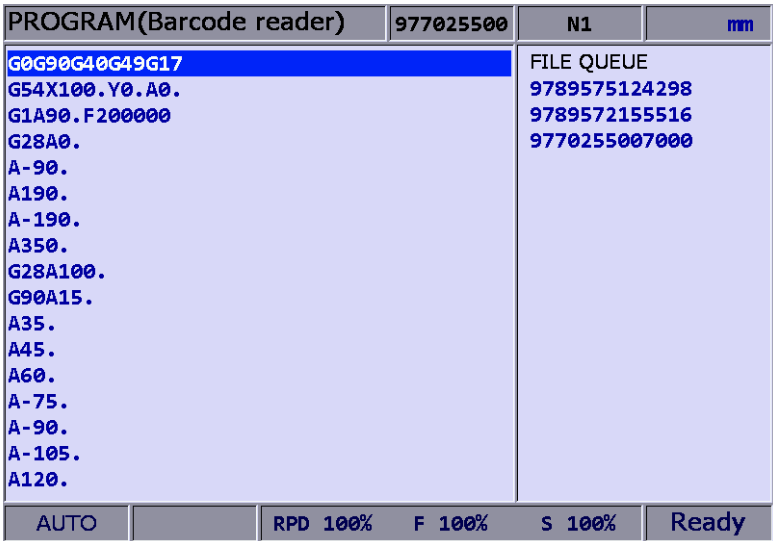


Figure 7.16.5

See the operation steps below:

1. In **AUTO Mode**, press the **PRG** key to enter the screen of [PROGRAM].
2. Press the **SCAN** to switch the screen for displaying. See figure 7.15.5.
3. Use the bar-code reader to acquire the machining filename.
4. Press the **LOAD** to load in the file content.
5. Or press the **CLR** to delete one file that displayed on the top of the scanning list.
6. Or you can press the **CLR ALL** to delete all files displayed from the list.

Note:

1. The machining file that will be loaded in the system by scanning its bar code should be created in CF card in advance. Also, its filename has to be identical to the bar code.
2. When multiple files are loaded, the system will execute each file in sequence. The system will delete the file from the list once its execution is complete. In addition, when only one file is loaded, the system will not delete it so that it can be executed repeatedly.

JOG and MPG feeding mode (JOG, MPG):

See the SF setting steps below:

1. In **JOG mode** or **MPG mode**, press the **PRG** key to enter the screen for program execution.
2. Press the **SF set** and the dialog box for SF command input will pop up.
3. Enter new S or F values. Then, press the **ENTER** key and the speed is changed.

Teach Programming: When users manually move the axis to any position, using function keys of teach programming can automatically convert the coordinates value of three axes into a motion command of one single block. This function shall be performed in JOG or MPG mode. The function of **Teach programming** is in PRG group, which can be operated in existing files or new files. Functions include rapid moving, linear cutting, arc cutting, deleting, creating files, saving files and selecting absolute / mechanical coordinates. It will automatically convert the file into the corresponded command format according to different functional selections. See below for the converting format.

Function	Auto generated command format
Create a new file when teach programming is enabled.	G90 G40 G49 G98 G50 G64 G80 G17 G69 G21 G54 G15 S3000 M03 F1000 ※According to the parameter (unit of length), it converts to G21 or G20 command.
Rapid moving	**G00** + X_Y_Z_
Linear cutting	**G01** + X_Y_Z_
Arc cutting	**G02** or **G03** + X_Y_Z_ + **I_ J_** ※According to plane X-Y, Z-X and Y-Z, it converts to G17+I_ J_, G18+K_ I_ or G19+J_K_.
Absolute coordinates	**G90** G00 (or G01/G02/G03) + X_Y_Z_
Mechanical coordinates	**G53** G00 (or G01/G02/G03) + X_Y_Z_

See the operation steps below for Teach programming:

1. Press the **PRG** key in **Jog mode** or **MPG mode** to enter the screen for performing the function.

2. Press the **TEACH** to enter the screen for teach programming.

3. Select the file and do teach programming in current file or new one. If desire to do programming in current file, users have to open the file in **Edit Mode**. If programming in a new file, press the **NEW FILE** to enter the file name in a pop-up input box. Then, press the **ENTER** key and users can create new files in current directory path.

4. Specify the data type of coordinates point. For example, to select the absolute coordinate, press the second toolbar and then press the **ABS**. Or press the **MECH** again to switch the data type to mechanical coordinates.

5. Move the axis to the specified position in **JOG mode** or **MPG mode**. Then, press the **RAPID** or the **LINEAR** according to the requirement of motion mode, which means to insert the coordinates command at the cursor position. And the coordinates command is generated based on the data type of its value.

6. To continue from Step (5), when it specifies arc motion, press the **ARC** to display the toolbar of acr cutting.

7. Then, specify arc plane setting. Press the **PLANE SEL** to select plane of X-Y, Y-Z or Z-X.

8. Move and setup the start point, middle point and end point of the arc in sequence by pressing **P1**, **P2** and **P3**. When the setting of P3 is complete, it is automatically converted into arc cutting command. The system will determine whether it is G02 or G03 and calculate its radius value then figure out the arc direction based on the sequence between P1 and P3.

9. If the coordinates command is incorrect, move the cursor to the block. Press the **DEL** in the first layer of toolbar in teach programming to delete the block.

10. When complete the operation of teach programming, apart from the auto-saving function (by pressing the **RESET** key, switching system mode, switching file), users can save the programming result by pressing the **SAVE**.

Note:

1. Teach programming has to be done in **Jog mode** or **MPG mode**; otherwise, the function will not be displayed.
2. The file size for teach programming is the same as file editing (under 3 MB).
3. For files created by teach programming, its filename has to comply with the naming rules.
4. When continuously input two same points, the second point will be ignored so as to avoid the ineffectiveness of motion block.
5. P1, P2, and P3 of arc command needs to be set up in sequence. Their positions determine the arc direction command and the distance of the circle center.
6. When the function of teach programming is enabled and no file is opened, the system will generate a blank file named “TEACH.NC” in the directory at the cursor position (Default: The file is generated in root directory of CF). Then, users may directly use the function of teach programming.
7. In SF setting, F command is enabled by the function of “Enable feed rate speed setting “ in parameter P10017.

Manual input mode (MDI):

The PRG group provides simple program entry, save, clear, and execution functions in manual mode. See the figure below for the program editing screen. This is exclusive to manual mode. Before the manually edited program is loaded in the system, cursor displays in a regular form which means the program is not running. Users can enter up to 17 lines of program steps. It is required to **load** the program again before running it. Otherwise, it cannot be executed.

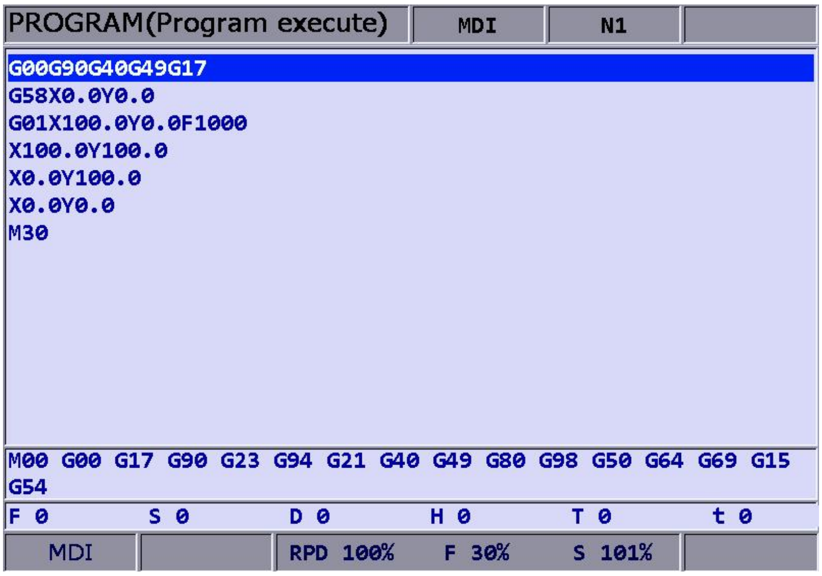


Figure 7.16.6

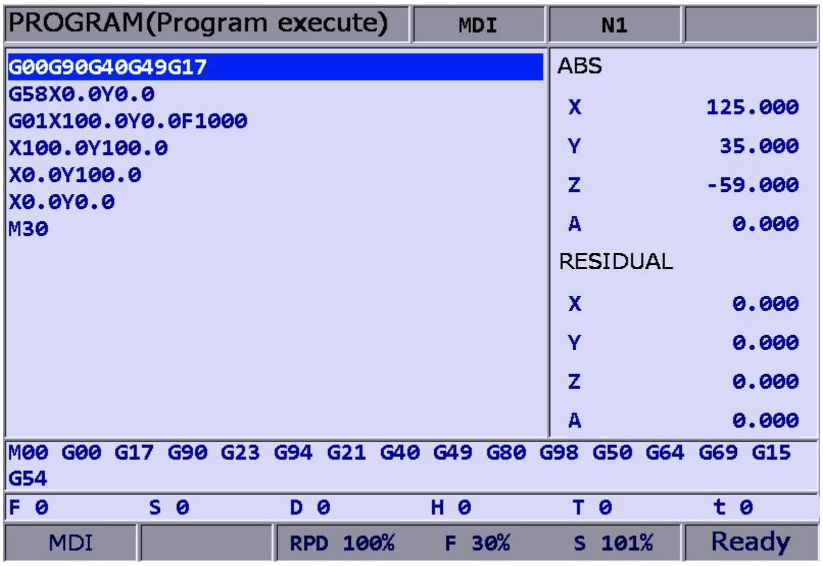


Figure 7.16.7

The file **save** function saves the manually edited file in the current directory following the same naming rule described in **Section 7.2 Create new file.** It requires giving a unique name in the current directory and with a format compliant with this standard. The **clear** function removes all contents in the programming page of manual mode. It functions the same as pressing and holds the **RESET** key for 3 seconds.

Note:

1. The **RESET** key has two functions in manual mode. The first one is the same as in auto mode which aborts the execution of a program and returns to the first line of a manual entry program. The second

can clear the contents in the manual entry area by pressing and holding the $\boxed{\textbf{RESET}}$ key for 3 seconds.

2. If M30 is executed and complete its execution in a block, the cursor will return to the first line, which is displayed as the line in execution.
3. In manual input mode, if the block has no M30 and when the program execution is complete, the cursor stops at the last line.
4. If the program's last block is M02, when its execution is complete, the cursor stops at the last line, which is displayed as the line in execution.

Offset (OFS) Group 8

The OFS group provides functions to set up workpiece coordinates, cutting tool length/ radius compensation, macros and variables.

Note: Here we use **Framed Text** to indicate the keys in primary control panel. And **boldface letter** is used for indicating the function key.

8

8.1 Coordinates setting

The coordinate system (G54 ~ G59) provides a function for setting multiple workpiece coordinate system. This enables users to designate coordinate data by working together with any workpiece coordinate system command in G54 ~ G59 of G code as illustrated in the figure below.

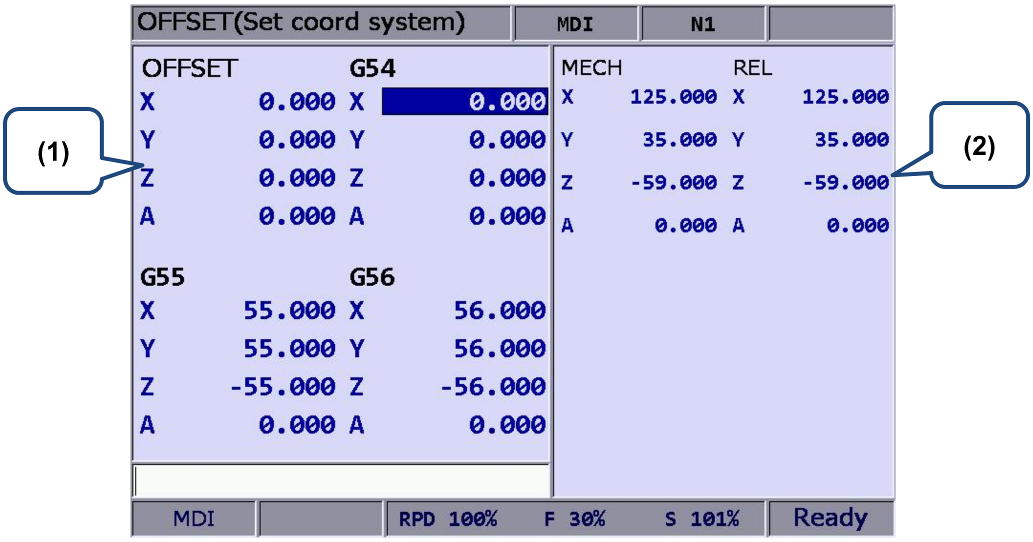


Figure 8.1.1

(1) Coordinate system setup: Offset coordinates/ G54 ~ G59
(2) Coordinates information: Mechanical coordinates / Relative coordinates

See the operation steps described below:

1. Press the **OFS** key to enter the screen of [Offset].
2. Press the **Coord** to enter the screen of coordinate system setup function bar.

Note:

1. Setting up coordinate system is only allowed when no machining program is executing. Otherwise, data entry will be rejected by the system.
2. End of execution of a single block is regarded as in non-running status while a paused program is in running status.

8.1.1 Auto setting

The auto setting function inputs the current position of each axis to the coordinate system (G54 ~ G59) where the cursor is. The input method includes single axis, multiple axis and L/2. The L/2 input shall work with the clear function of relevant axis. The auto setting function also clears the numeric values of a given coordinates system with sub-functions of relative clear, all clear, L input, L/2 input and P input.

■ **All Clear:** It clears all axes values of the current coordinate system to zero while all other coordinate systems' values remain unchanged.

See the operation steps below:

1. Press the **OFS** key to enter the screen of [Offset].
2. Press the **Coord** to enter the screen of coordinate system setup function bar.
3. Press the **Auto** to switch to the screen with the coordinates auto setup function bar.
4. Use ↑, ↓, ←, and → keys to move the cursor to the position of the specified coordinate system group.
5. Press the **CLR ALL** to remove all coordinates group data where the cursor resides.

■ **Relative clear:** This function clears the relative coordinates that correspond to the cursor position. Axis types are determined by cursor position, while those irrelevant to the cursor remain unchanged. This function clears relative coordinate value in the coordinates display rather than the data of the actual workpiece coordinate system.

■ **L/2 input:** When identifying the center of an object and to regard this center as the origin of the coordinate system, this function is able to automatically figure out the coordinate data and complete the setting. See the operation steps below (illustration based on X-axis):

1. In [**Jog mode**] or [**MPG mode**], move the mechanism to the X-axis of the workpiece coordinate and regard the first contact point as the origin on the X-axis.
2. Press the **OFS** key to enter the screen of [Offset].
3. Press the **Coord** to enter the screen of coordinate system setup function bar.
4. Press the **Auto** to switch to the screen with the coordinates auto setup function bar.
5. Use ↑, ↓, ←, and → keys to move the cursor to the X-axis coordinates position of the specified coordinate system group.
6. Press the **SET L/2** to enter the L/2 input screen.
7. Press the **Point1**. See Figure 8.1.1.1, the first circle on the rectangle turns red; this means the recording of the first mechanical coordinate value is complete.

8

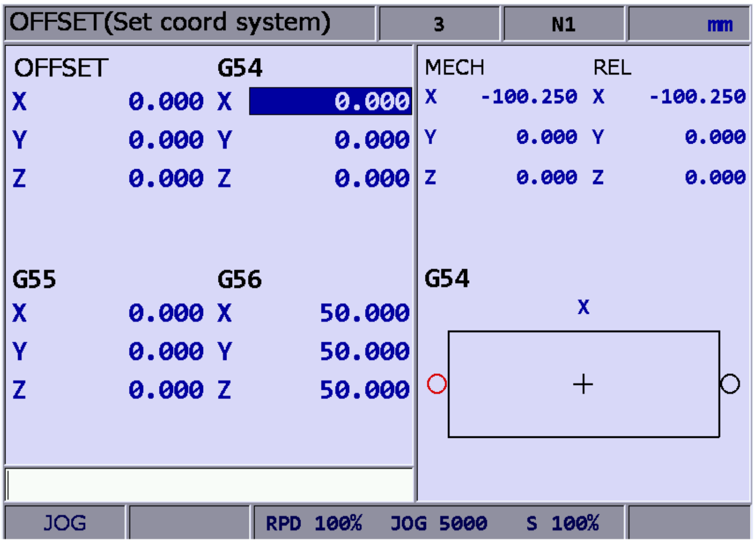


Figure 8.1.1.1

8. Move the mechanism to the contact point on the other side of the X-axis.
9. Press the **Point2**. See Figure 8.1.1.1. When the second circle turns red, it means the mechanical coordinate value of the second point has been recorded.
10. Press the **SET**. The system will calculate the origin on the X-axis of the workpiece coordinates. It will measure the distance from the mechanical origin to the end of the X-axis of the workpiece coordinates. This value times 0.5 will be the center on the X-axis of this coordinates.

- **L input:** This function automatically inputs the current mechanical coordinate value to the field that corresponds to the cursor position. This function inputs single-axis coordinate data only. See the operation steps described below:

1. In [**Jog mode**] or [**MPG mode**], move the mechanism to the first contact point on the X-axis of the workpiece coordinates.
2. Press the **OFS** key to enter the screen of [Offset].
3. Press the **Coord** to enter the screen of coordinate system setup function bar.
4. Press the **Auto** to switch to the screen with the coordinates auto setup function bar.
5. Use ↑, ↓, ←, and → keys to move the cursor to the X-axis coordinates position of the specified coordinate system group.
6. Press the **SET L** and the axis coordinate value input from the highlighted part is complete.

Example of L input (for the X-axis):

Move the mechanism to the specified position in the coordinate system, as with the origin on the X-axis of the workpiece coordinates in Figure 8.1.1.2.

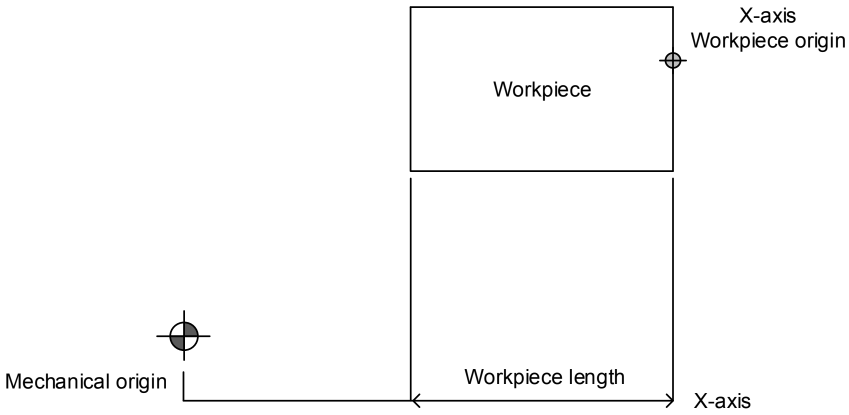


Figure 8.1.1.2

The mechanical coordinate data is shown in mechanical coordinate section. Move the cursor to the specified coordinate group, such as G54 in Figure 8.1.1.3. Then, press **SET L** and the X-axis data of the mechanical coordinate is inputted to the X-axis fields of the G54 automatically. The single axis data input of the coordinate group is now completed.

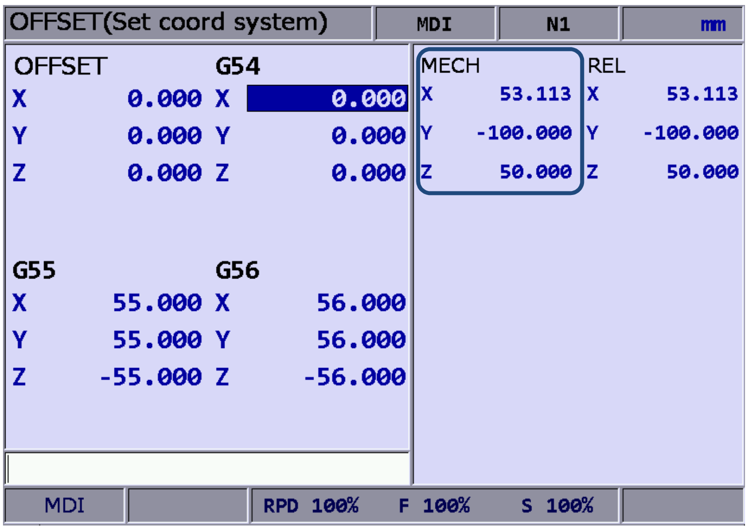


Figure 8.1.1.3

8

■ **P input:** This function inputs the coordinate center of multiple axes concurrently after the workpiece center point is calibrated. With P input function, more than one axis, including X-, Y-, and Z-axis can be inputted.

See the operation steps below:

1. In [**Jog mode**] or [**MPG mode**], move the mechanism to the initial contact point on the X-axis of the workpiece coordinates.
2. Press the **OFS** key to enter the screen of [Offset].
3. Press the **Coord** to enter the screen of coordinate system setup function bar.
4. Press the **Auto** to switch to the screen with the coordinates auto setup function bar.
5. Use ↑, ↓, ←, and → keys to move the cursor to the data position of the coordinate system group.
6. Press the **SET P**, multiple axis data is now automatically inputted into the highlighted coordinate group fields.

Note: Do not press **All Clear** function key to clear the coordinate value, or it would clear the coordinate values of all other axes that have been set.

Example of P input:

Move the mechanism to the specified coordinate position, e.g. the workpiece origin in Figure 8.1.1.4. (Figure 8.1.1.4 indicates the relative position of the X- and Y-axis but not the Z-axis.)

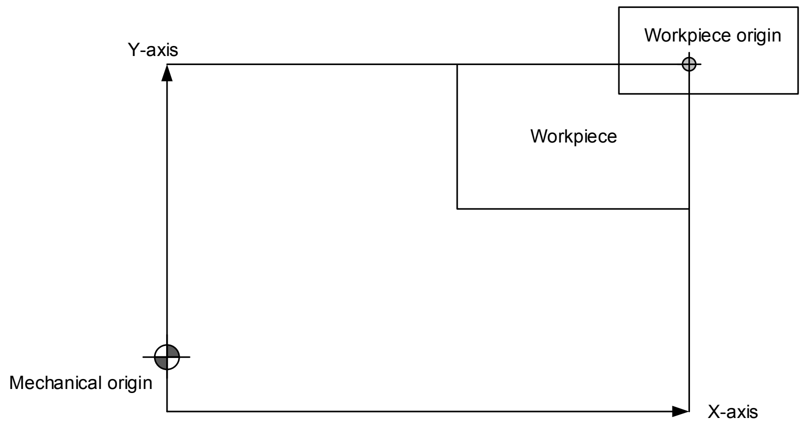


Figure 8.1.1.4

After the workpiece origin is calibrated, the mechanical coordinate data is shown in the mechanical coordinate fields in Figure 8.1.1.5. Move the cursor to the specified coordinates group (e.g. the G54 group in Figure 8.1.1.5). Press **SET P**, and then the X-, Y-, and Z-axis data of the mechanical coordinate are inputted to the mechanical coordinate fields of G54. That is, the multiple axis data input for the axis group is completed.

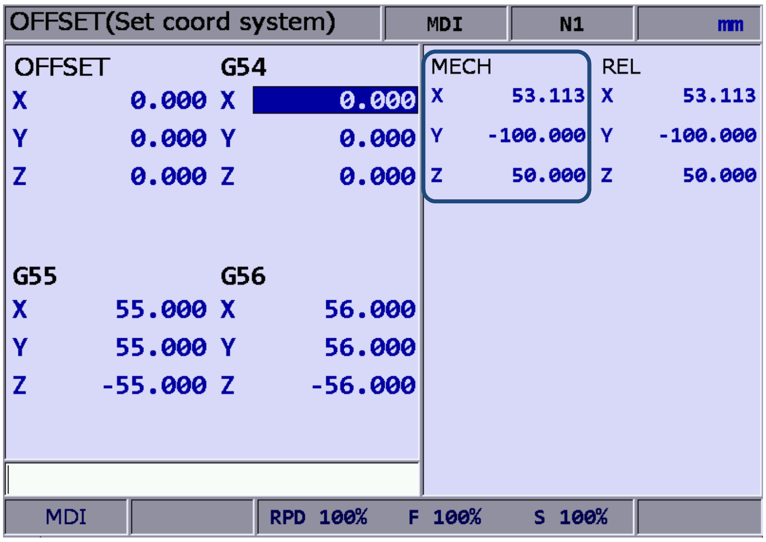


Figure 8.1.1.5

8.1.2 Absolute input

The value of coordinate system can be inputted manually by absolute or incremental value setups. This section explains the steps for absolute input.

See the operation steps below:

1. Press the **OFS** key to enter the screen of [Offset].
2. Press the **Coord** to enter the screen with coordinate system setup function bar.
3. Use , , , and keys to move the cursor to the X-, Y-, and Z-axis setup positions of the specified coordinate system.
4. Input positive or negative values by pressing keys **0**~**9.** To enter a negative value, press the key in advance. Press the key to confirm the unit of values.
5. Press the **ABS** to enter the value of the coordinate.

Note:

1. The unit of value is mm. Value without decimal points is in unit of μm. That is, input value 123456 indicates 123.456 mm.
2. The absolute input can be made by step (5) as described above or by pressing the **ENTER** key.

Example of absolute input:

Move the tool center from mechanical origin to the origin of the workpeice coordinates. Next, input this coordinate value (X and Y) to the controller's **OFS** group (G54 ~ G59). Then, execute the corresponded command in the G code program and the setting for origin of the workpiece coordinates is complete.

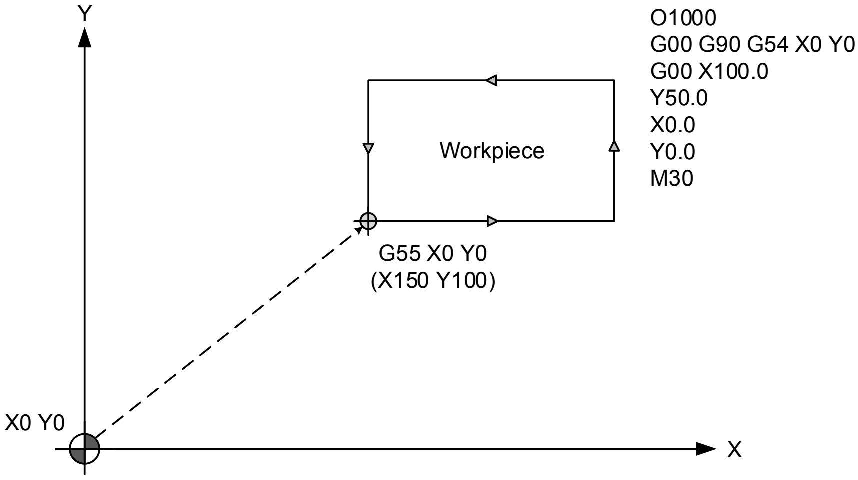


Figure 8.1.2.1

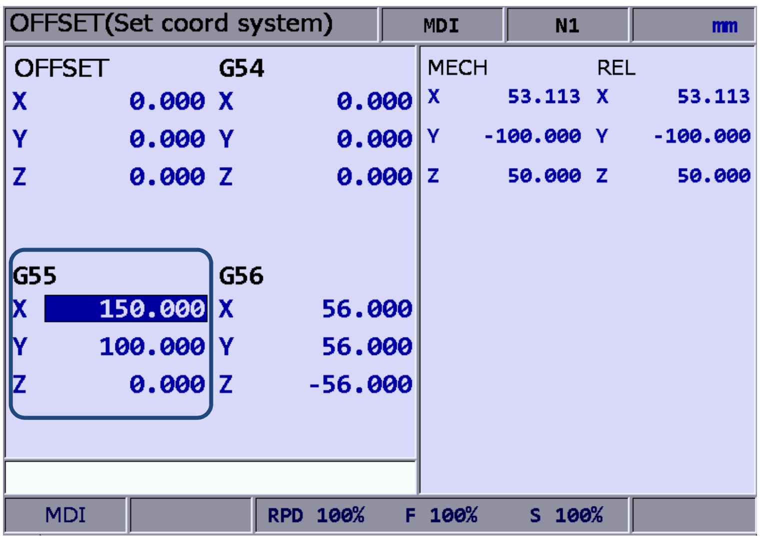


Figure 8.1.2.2

8.1.3 Incremental input

This is one of the manual methods for inputting coordinate data. Generally, incremental input is applied for fine tuning as the value is input incrementally. For example, if the original value is 150.000, with an incremental input of 5.000, the new value shall be 155.000.
See the operation steps below:

1. Press the **OFS** key to enter the screen of [Offset].
2. Press the **Coord** to enter the screen with coordinate system setup function bar.
3. Use , , , and keys to move the cursor to the X-, Y-, and Z- axis setup position of the specified coordinate system.
4. Input positive or negative values by pressing keys **0** ~ **9.** To enter a negative value, press the key in advance. Then, press the key to confirm the unit of value.
5. Press the **INC** to incrementally increase or decrease the axis position value.

Note: When manually inputting the data, please make sure the value and the inputting method is correct so as to avoid the danger caused by any incorrect movement.

8.1.4 Rectangle center

This function assists users in setting up coordinate value of the rectangle center with a rectangle drawing as shown in the figure below. The system converts the data of the four corners into coordinate value of the object's actual center.

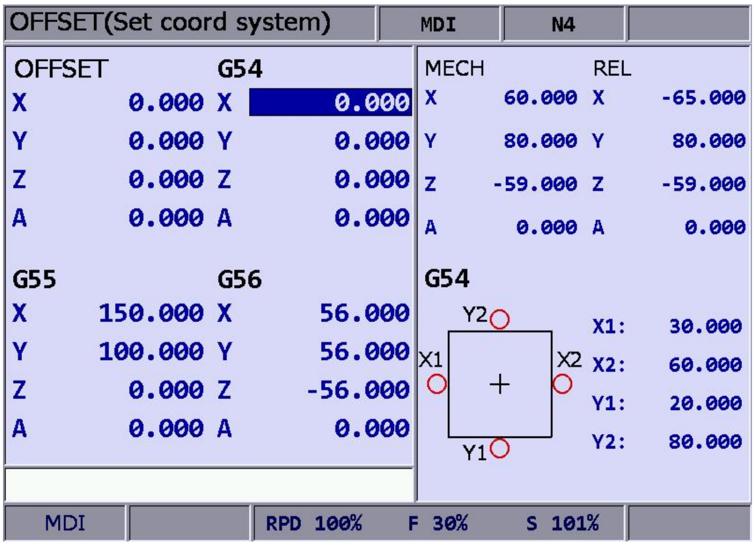


Figure 8.1.4.1

See the operation steps below:

1. Press the $\boxed{\text{OFS}}$ key to enter the screen of [Offset].
2. Press the **Coord** to enter the screen with coordinate system setup function bar.
3. Use ↑, ↓, ←, and → keys to move the cursor to the data field of the specified coordinate system.
4. Press the **SQUARE** to enter the screen of rectangle center.
5. As guided by the rectangle shown in the screen, move the spindle center to the mechanical position of X1, X2, Y1, and Y2. And press **X1**, **X2**, **Y1**, and **Y2** keys to set up the coordinates data of each point.
6. Press the **Set** after coordinates of the four points are set, the system will calculate the coordinate data of the rectangle center and input the data to the coordinate system.
7. Set up the coordinate position by moving the Z-axis, press the **SET Z** to set up the Z-axis coordinates of the workpiece coordinate group.

Example of rectangle center:

Firstly specify the fields of coordinates group. Then, manually move the spindle to the four corners of the object, enter the X and Y coordinates data of these 4 points as shown in Figure 8.1.4.2.

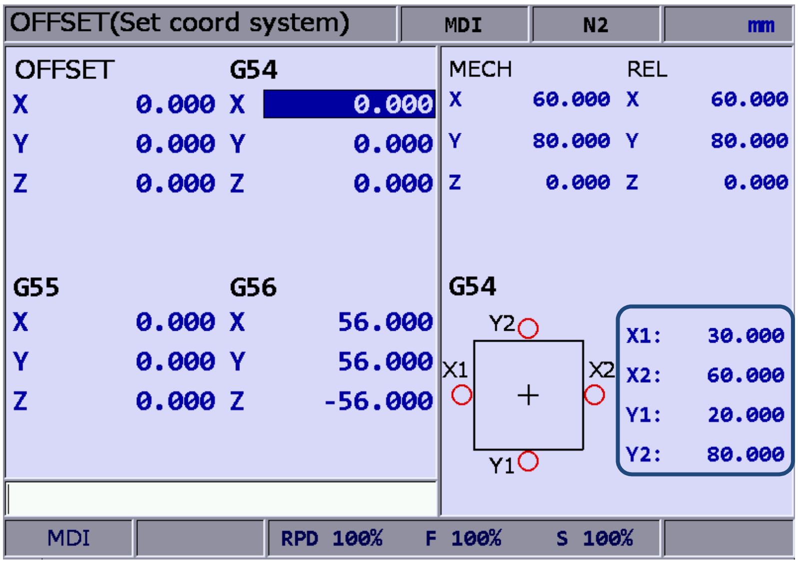


Figure 8.1.4.2

After the coordinates of the four rectangle corner points are set, press the **Set**; the system will then figure out the actual mechanical coordinates value of the rectangle object center and set up given coordinates system data as shown in Figure 8.1.4.3.

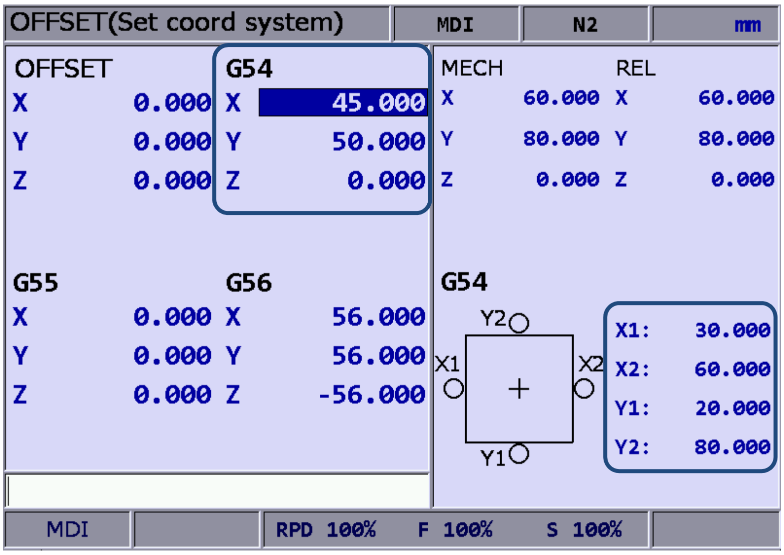


Figure 8.1.4.3

8

8.1.5 Circle center

This function sets up the coordinates data of the center of any ball object. Select any 3 points on the ball object and set up their coordinates data; this function can figure out the coordinates of the object center as shown in the figure below:

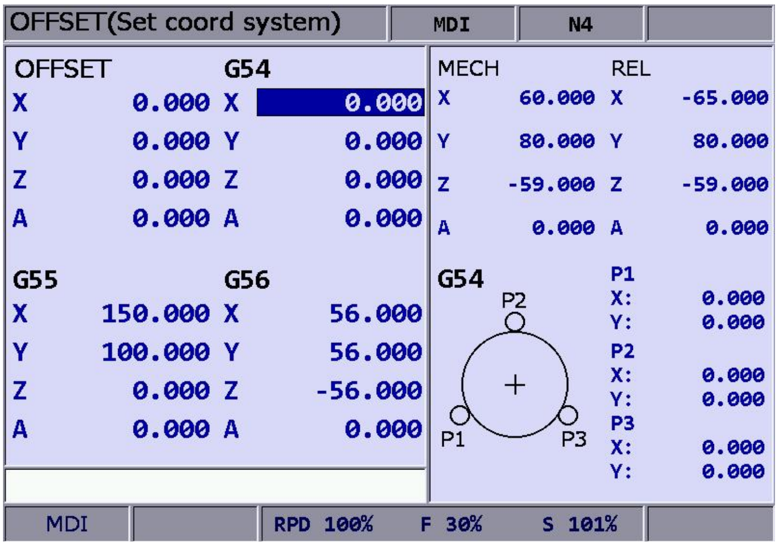


Figure 8.1.5.1

See the operation steps below:

1. Press the OFS key to enter the screen of [Offset].
2. Press the **Coord** to enter the screen with coordinate system setup function bar.
3. Use ↑, ↓, ←, and → keys to move the cursor to data fields of the specified coordinate system.
4. Press the **CIRCLE** to enter the screen for circle center.
5. Move the mechanical position of P1, P2, and P3 as guided. Then, press **P1**, **P2**, and **P3** keys to set up the position one by one.
6. After setting up coordinates of the three points on a circle, press the **SET**; the system will automatically figure out the coordinates data of the circle center and input the data to the coordinate system.
7. Set up coordinate position by moving the Z-axis, press the **SET Z** to set up the Z-axis coordinates of the workpiece coordinates group.

Example of circle center:

This function applies to any workpiece in a ball object. Manually move the spindle to access any three outer points on the circle, and then set up the coordinate value of these three points by relevant function keys as shown in Figure 8.1.5.2.

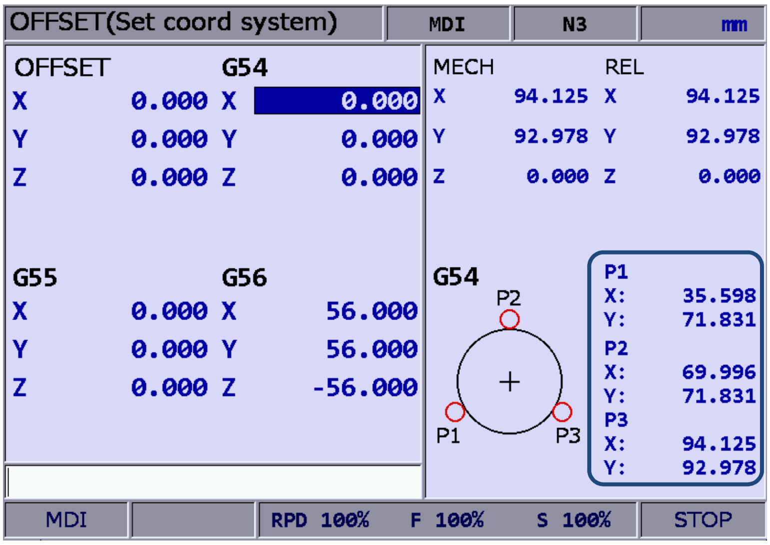


Figure 8.1.5.2

After the coordinates of any three outer points are set, press the **Set**, the system will then automatically figure out the actual mechanical coordinates value of the circle center and sets up given coordinates system data as shown in Figure 8.1.5.3.

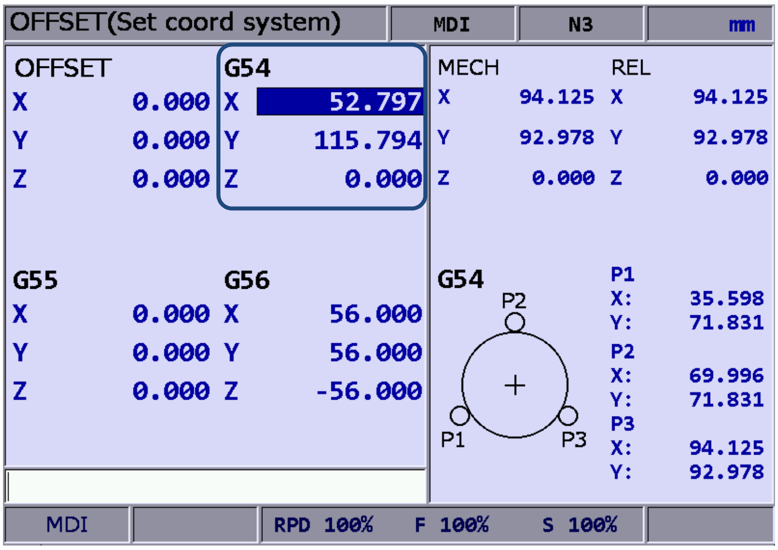


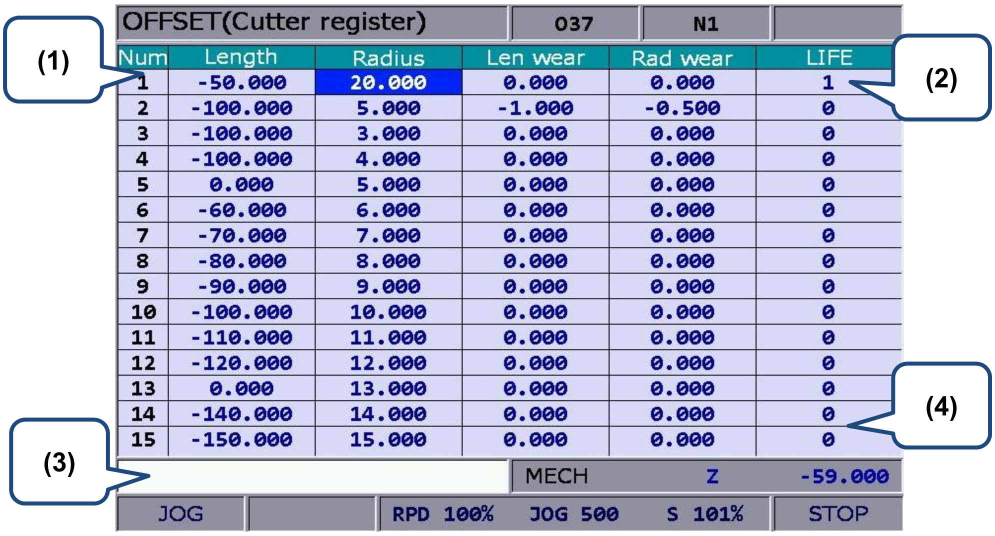
Figure 8.1.5.3

8.2 Tool register

8.2.1 Milling System

This function varies with tool length compensation (G43 or G44, or cancel command G49) or radius compensation (G41 or G42, or cancel command G40). The tool register function covers tool length compensation, radius compensation, length wear compensation, and radius wear compensation.

The data fields correspond to **H (tool length compensation)** and **D (tool radius compensation)** codes assigned by the machining program. See the figure below for the tool compensation function screen.



Num	Length	Radius	Len wear	Rad wear	LIFE
1	-50.000	20.000	0.000	0.000	1
2	-100.000	5.000	-1.000	-0.500	0
3	-100.000	3.000	0.000	0.000	0
4	-100.000	4.000	0.000	0.000	0
5	0.000	5.000	0.000	0.000	0
6	-60.000	6.000	0.000	0.000	0
7	-70.000	7.000	0.000	0.000	0
8	-80.000	8.000	0.000	0.000	0
9	-90.000	9.000	0.000	0.000	0
10	-100.000	10.000	0.000	0.000	0
11	-110.000	11.000	0.000	0.000	0
12	-120.000	12.000	0.000	0.000	0
13	0.000	13.000	0.000	0.000	0
14	-140.000	14.000	0.000	0.000	0
15	-150.000	15.000	0.000	0.000	0

Figure 8.2.1

(1) Compensation code (H/D)

(2) Compensation Data: Tool length; Tool radius; Length compensation; Radius compensation

(3) Compensation data input field

(4) Auxiliary display: Mechanical coordinates and actual position of current Z-axis

Range of tool register values	
Range of tool length data	-2000.0 ~ 2000.0 mm
Range of tool radius data	-150.0 ~ 150.0 mm
Range of tool length wear compensation data	-2000.0 ~ 2000.0 mm
Range of tool radius wear compensation data	-150.0 ~ 150.0 mm
Range of tool life span	0 ~ 99999999 serves

■ **Absolute input:** This is one of the manual data input methods. Use this function to input absolute values of tool length, tool radius, wear compensation or tool life span data. Absolute value input can also be done by pressing the **ENTER** key.

See the operation steps below:

1. Press the **OFS** key to enter the screen of [Offset].
2. Press the **CUTTER** to enter the screen with the tool register function bar.
3. Use ⇑, ⇓, ⇐, and ⇒ keys to move the cursor to data fields for tool length, radius or wear designation.
4. Input positive or negative values by pressing keys **0** ~ **9.** To enter a negative value, press the key in advance. Press the key before entering the tool compensation data to ensure the unit of value. Only positive integers are valid input for tool life span.
5. Press the **ABS** to register absolute values.

Note: The tool data fields are for individual compensation values. For example, when the length fields are highlighted, it means the input data is for tool length compensation.

■ **Incremental input:** This is one of the manual data input methods. Use this function to input incremental values of tool length, tool radius, wear compensation or tool life span data. See the operation steps below:

1. Press the key to enter the screen of [Offset].
2. Press the **CUTTER** to enter the screen with the tool register function bar.
3. Use ⇑, ⇓, ⇐, and ⇒ keys to move the cursor to data fields for tool length, radius, wear, or life span designation.
4. Input positive or negative values by pressing keys **0** ~ **9.** To enter a negative value, press the key in advance. Press the key before entering the tool compensation data to ensure the unit of value. Only positive integers are valid input for tool life span.
5. Press the **INC** to register incremental values.

■ **H Setup:** This function automatically inputs the height of Z-axis of current mechanical coordinates in the assigned tool length compensation data field (H).

See the operation steps below:

1. In [**Jog feeding mode**] or [**MPG feeding mode**], move the Z-axis to specified coordinates height.
2. Press the **OFS** key to enter the screen of [Offset].
3. Press the **CUTTER** to enter the screen with the tool register function bar.
4. Use ↑, ↓, ←, and → keys to move the cursor to the tool length fields for tool code designation.
5. Press the **SET H** to set the current Z-axis mechanical coordinates value in the given fields.

Note:

1. The H setup function applies to tool length data fields only.
2. Do not change values in OFS group during program execution. Enter values only when the program stops. The program stop status means the program is not in operation, a block is completed when single-block stop function is enabled, or after the RESET key is pressed.
3. The length wear value is reset to zero when inputting tool length value with H setup.

■ **Clear:** This function clears tool compensation values with options of geometry, wear, life span, and all clear.

Geometry clear: clear all tool length and radius values.

Wear clear: clear all tool length compensation and radius compensation values.

Life span clear: clear all tool life span values.

All clear: clear all tool registry data.

See the operation steps below:

1. Press the **OFS** key to enter the screen of [Offset].
2. Press the **CUTTER** to enter the screen with tool register function bar.
3. Press the **Clear** to display clear function bar.
4. Press the **H/D** to clear tool length and radius values. Press **Wear** to clear all tool length compensation and radius compensation values. Press the **Life** to clear all tool life span values. Press **All** to clear all tool registry data.

8.2.2 Turning System

■ When programming, you have to specify the radius compensation number in the program. These numbers all correspond to the number in the compensation data table. And the radius compensation values are inputted via the tool register function via OFS group by users in advance. The tool register function of the turning system includes: tool length compensation, tool length wear compensation, tool nose radius, radius wear, and tool nose type.

■ Format of tool compensation:

T0204: 02 at the front represents tool number; 04 represents tool length compensation and tool wear compensation

T02: When it only contains one set of number, the number stands for the tool number, tool length compensation and tool wear at the same time, which can also be presented as T0202.

See the following figure for tool length setting:

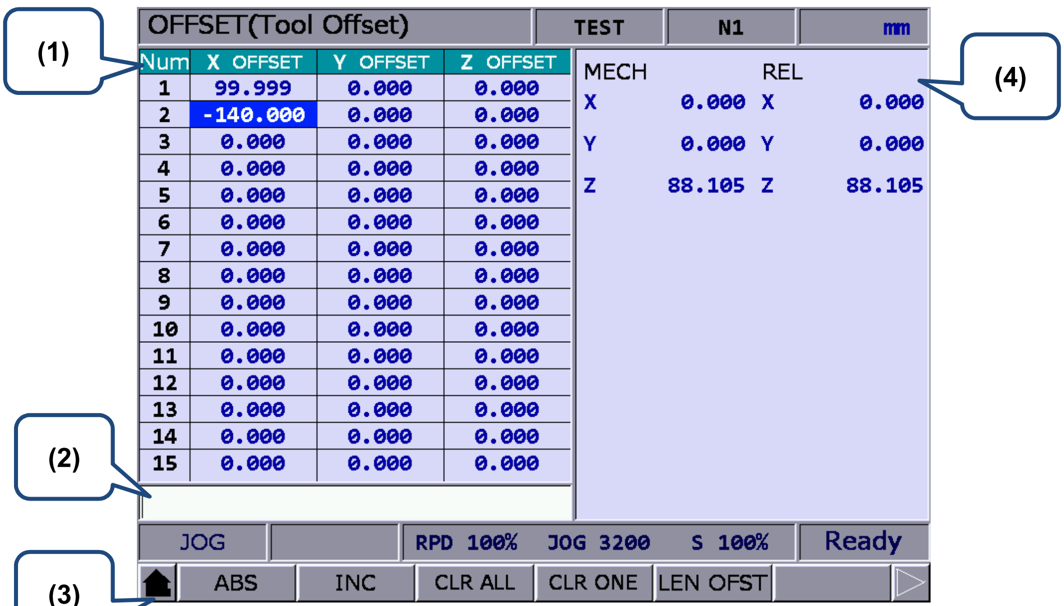


Figure 8.2.2.1

(1) Compensation number and corresponding tool length

(2) Input field of compensation data

(3) Input mode selection

(4) Auxiliary display: mechanical coordinates and relative coordinates

Data range of tool register	
Data range of tool register	-2000.0 ~ 2000.0 mm

8

See the following figure for tool wear setting

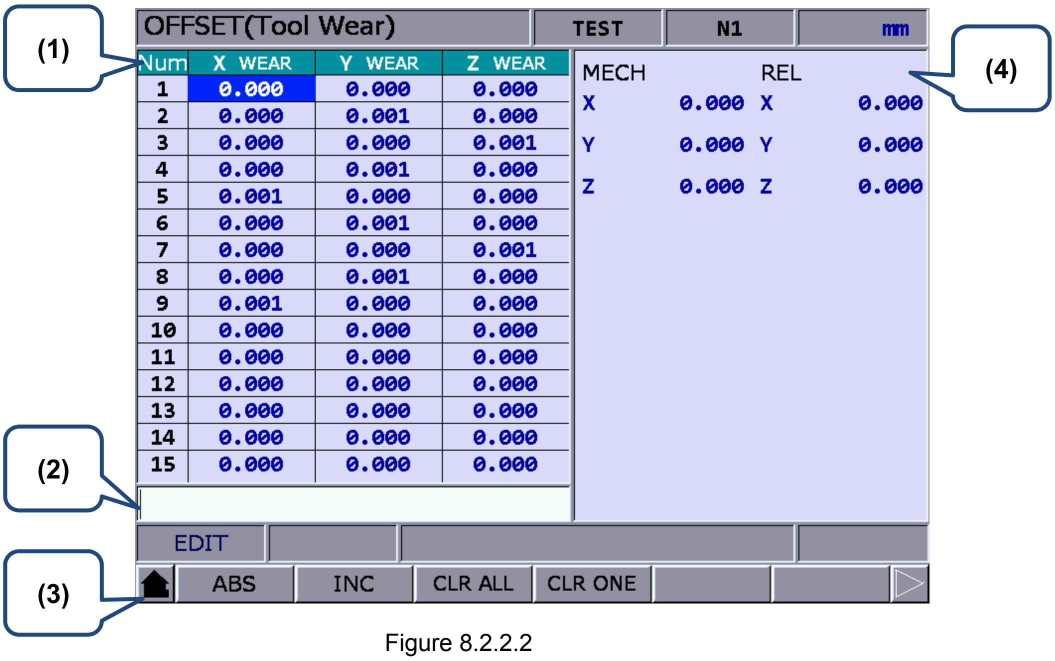


Figure 8.2.2.2

(1) Tool wear compensation number and corresponding wear value of the axis

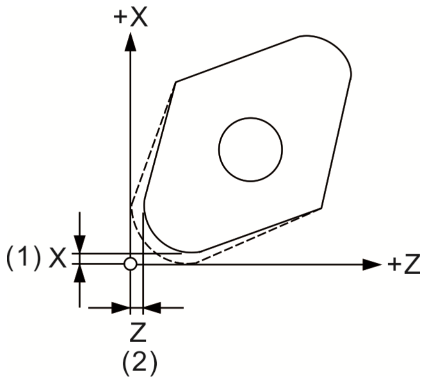
(2) Input field of compensation data

(3) Input mode selection

(4) Auxiliary display: mechanical coordinates and relative coordinates

Data range of tool register	
Wear data range of each axis	-2000.0 ~ 2000.0 mm

Tool wear:



(1) Tool nose wear compensation amount of axis X;

(2) Tool nose wear compensation amount of axis Z

See the following figure for tool nose setting:

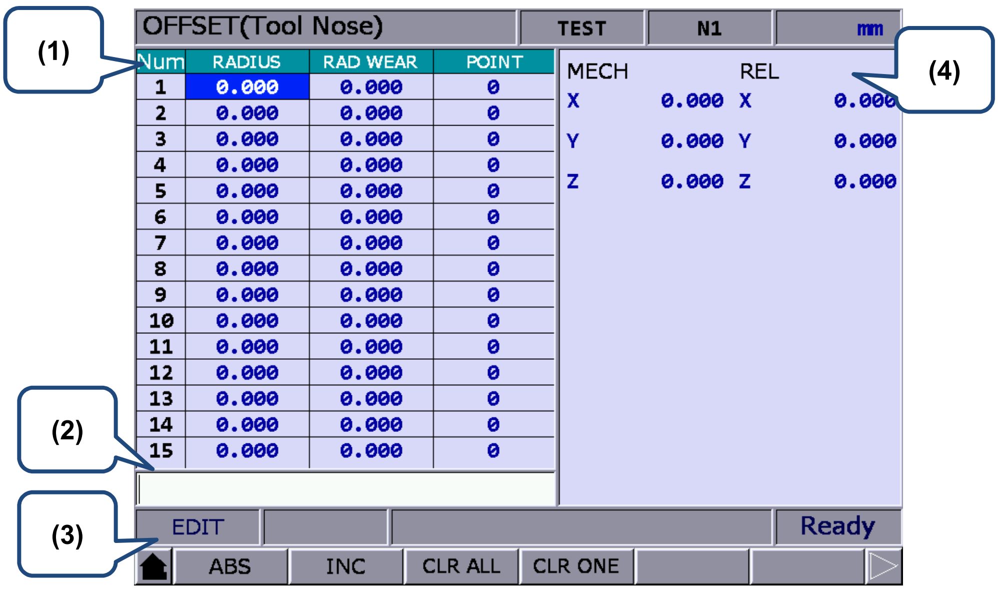


Figure 8.2.2.3

(1) Tool nose compensation number and corresponding radius compensation value

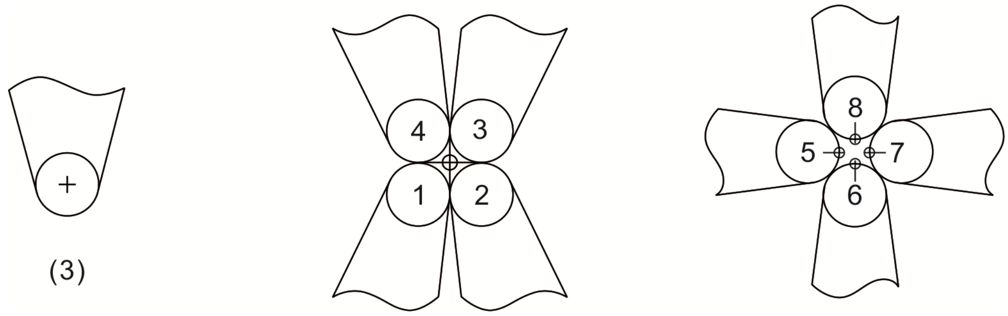
(2) Input field of compensation data

(3) Input mode selection

(4) Auxiliary display: mechanical coordinates and relative coordinates

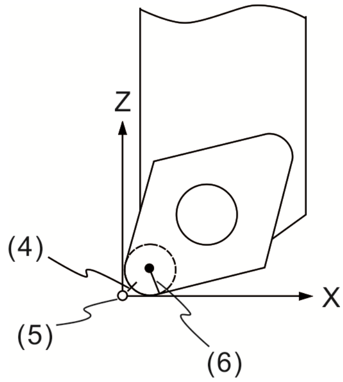
Data range of tool register	
Data range of tool nose radius compensation	-2000.0 ~ 2000.0 mm
Data range of radius wear compensation	-2000.0 ~ 2000.0 mm
Data range of tool nose type	0 ~ 9

Tool nose type:



(3) Tool nose 0 or 9

Tool nose and tool nose radius compensation:



(4) Actual tool nose; (5) The assumed tool nose position when calibration;

(6) Tool nose radius compensation R

- Absolute input **ABS**: When using this function, the data is inputted manually. Tool length, tool radius, wear compensation and tool life data can all be inputted in absolute form. You can also press the **ENTER** key to do the setting.

- Incremental input **INC**: When using this function, the data is inputted manually. Tool length, tool radius, wear compensation, tool life data can all be inputted in incremental form.

- Clear all **CLR ALL**: Clear all compensation data in the whole page.

- Clear single axis **CLR ONE**: Clear the compensation data of one axis.

- Tool length offset **LEN OFST**: This function is for auto inputting tool length compensation only. You can input the absolute position of each axis and get the corresponding tool length compensation of each axis automatically. Unlike manually inputting the values, this function will help you input the correct value and save the setting time.

See the following operation steps:

1. Use ⬆, ⬇, ⬅, and ➡ keys to move the cursor to the tool length field of the specified number.
2. Input the absolute coordinates in the input field and press **Offset**, the controller will automatically calculate the tool length compensation amount of the axis where the cursor specifies.

The calculation will be **[Current mechanical coordinates] – [The inputted absolute coordinates] = [Tool length compensation value]**

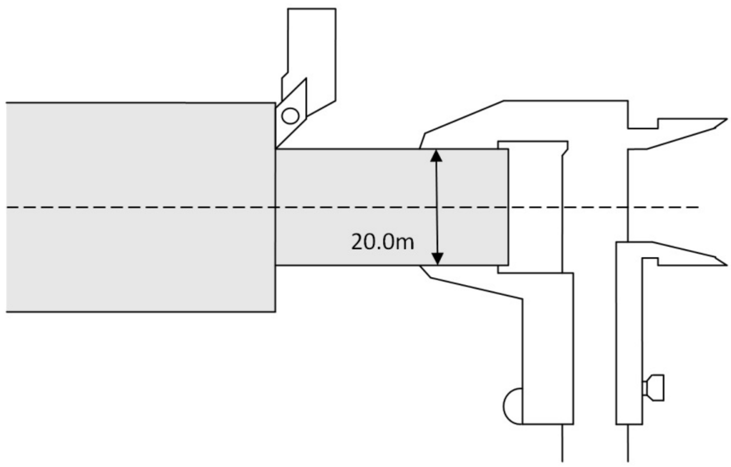
[The inputted absolute coordinates] is a negative value.

[Tool length compensation of axis X]

Input 20.0 mm (the measured diameter of X-axis after cutting) to the compensation field of X-axis.

Then, press the **Offset** key and you will get the tool length compensation value automatically.

Note: When cutting is completed and the **Offset** key is pressed, do not move axis X. (Do not change its mechanical coordinates)



8

8.3 Tool magazine register

This function manages the tool positions corresponding to the tool magazine after tool exchanges. The tool magazine data is a table recording the actual tool pot positions and tool ID of the machine. It not only records and displays the tool pot position of individual tools but also changes the tool sequence in tool magazine data fields with permission. **The tool magazine register function runs in Jog feeding mode only.** See the figure below.

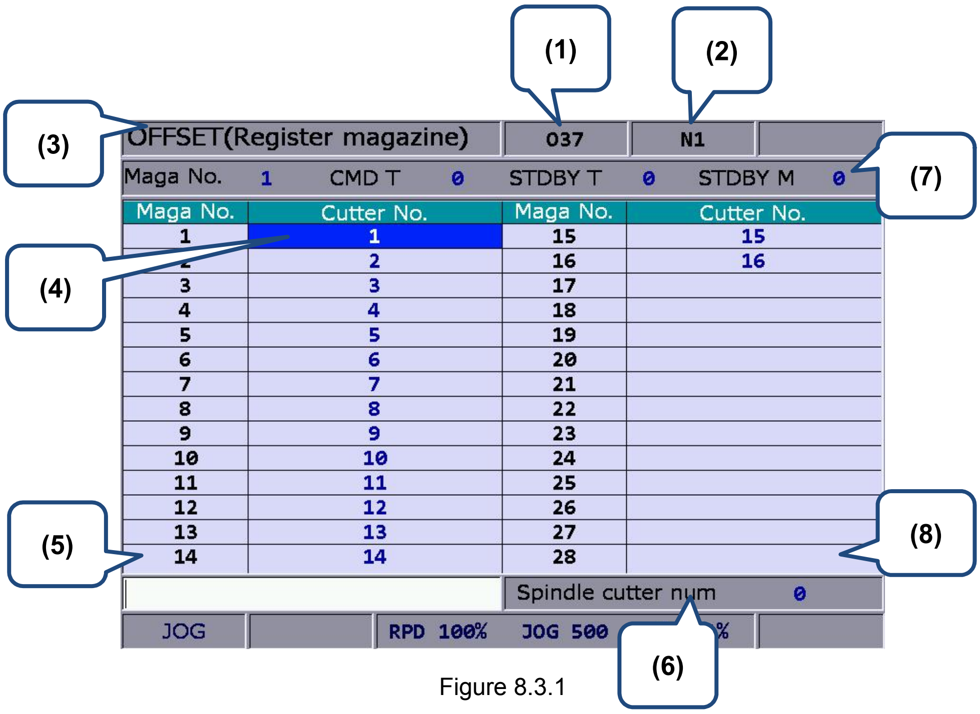


Figure 8.3.1

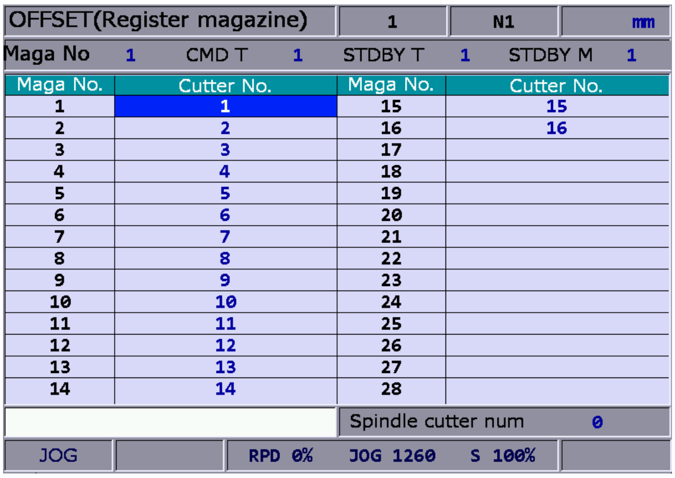
(1) Current program
(2) The line being executed
(3) Current group screen
(4) The sequential status of tool
(5) Tool ID input field
(6) Rate of current motion
(7) Maga No.: Tool magazine system number
CMD T: Current command tool ID
STDBY T: Current standby tool ID
STDBY M: Current standby tool pot ID
(8) Spindle tool ID

See the operation steps below for tool magazine setup:

1. Set the system to **Jog feeding mode**.
2. Press the **OFS** key to enter the screen of [Offset].
3. Press the **MAGA** to enter the tool magazine data setup function screen.
4. Use ⬆, ⬇, ⬅, and ➡ keys to move the cursor to the assigned data fields.
5. Enter the newly changed tool ID, press the **SET** (or press the **ENTER** key) to set up the position of new tool magazine.

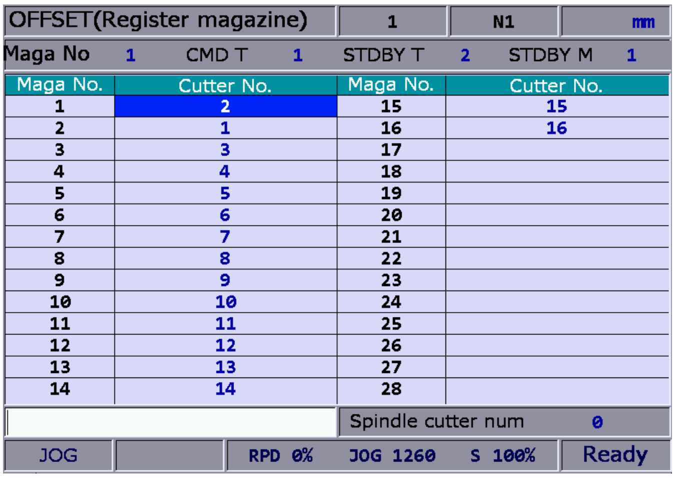
Example of tool ID exchange:

If the assigned tool ID duplicates one in existence, then it exchanges it with the one at the original place automatically. This ensures that each tool ID in the tool magazine does not duplicate another and prevents incorrect tool calling.



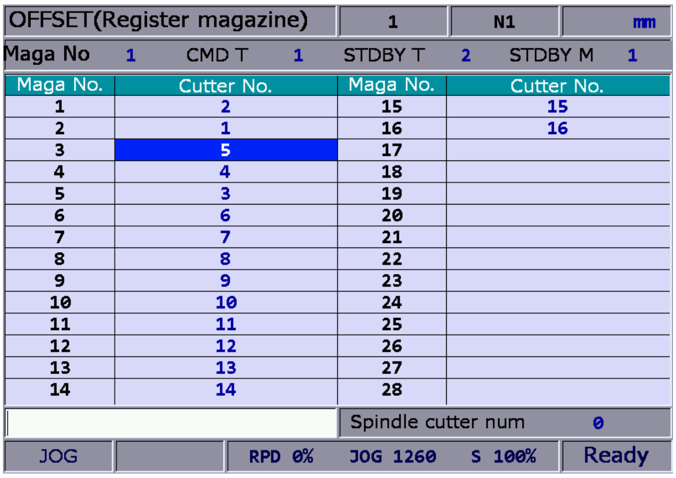
Maga No.	Cutter No.	Maga No.	Cutter No.
1	1	15	15
2	2	16	16
3	3	17	
4	4	18	
5	5	19	
6	6	20	
7	7	21	
8	8	22	
9	9	23	
10	10	24	
11	11	25	
12	12	26	
13	13	27	
14	14	28	

Initial status of the tool magazine with tools in it in numeric sequence



Maga No.	Cutter No.	Maga No.	Cutter No.
1	2	15	15
2	1	16	16
3	3	17	
4	4	18	
5	5	19	
6	6	20	
7	7	21	
8	8	22	
9	9	23	
10	10	24	
11	11	25	
12	12	26	
13	13	27	
14	14	28	

Set position 1 = 2, and tools ID 1 and 2 in tool magazine 1 and 2 exchanges with each other.



Maga No.	Cutter No.	Maga No.	Cutter No.
1	2	15	15
2	1	16	16
3	5	17	
4	4	18	
5	3	19	
6	6	20	
7	7	21	
8	8	22	
9	9	23	
10	10	24	
11	11	25	
12	12	26	
13	13	27	
14	14	28	

Set position 3 = 5, and tools ID 3 and 5 in tool magazine 3 and 5 exchanges with each other.

This demonstrates that tool IDs in the tool magazine exchange with each other after the tool ID of a given tool magazine number is changed. This eliminates errors caused by invalid tool ID accessing.

■ **All reset:** The tool register also provides the reset function of tool magazine position. This resets the tool ID in the tool magazine to default. This function can be used for misplacement troubleshooting or tool ID reset.

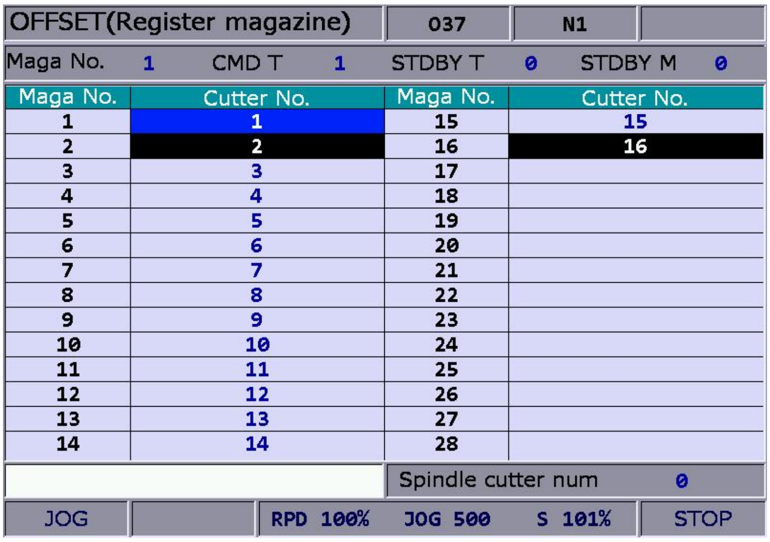
See the operation steps below:

1. Set the system to **Jog feeding mode**.
2. Press the **OFS** key to enter the screen of [Offset].
3. Press the **MAGA** to enter the screen of tool magazine data setup function.
4. Press the **RST ALL** to reset all tool magazine position records.

■ **Tool magazine block:** This function blocks the tool magazine position not used by the program. Tools in a blocked tool magazine position cannot be called. If they are called incorrectly, the system blocks their use, warning with an error message, and will halt program execution immediately. A blocked tool magazine position is identified by a different color.

See the operation steps below:

1. Set the system to **Jog feeding mode**.
2. Press the **OFS** key to enter the screen of [Offset].
3. Press the **MAGA** to enter the screen of tool magazine data setup function.
4. Use , , , and key to move the cursor to the assigned data fields.
5. Press the **LOCK** to block the tool magazine position as shown in Figure 8.3.2.


OFFSET(Register magazine) 037 N1

Maga No. 1 CMD T 1 STDBY T 0 STDBY M 0

Maga No.	Cutter No.	Maga No.	Cutter No.
1	1	15	15
2	2	16	16
3	3	17	
4	4	18	
5	5	19	
6	6	20	
7	7	21	
8	8	22	
9	9	23	
10	10	24	
11	11	25	
12	12	26	
13	13	27	
14	14	28	

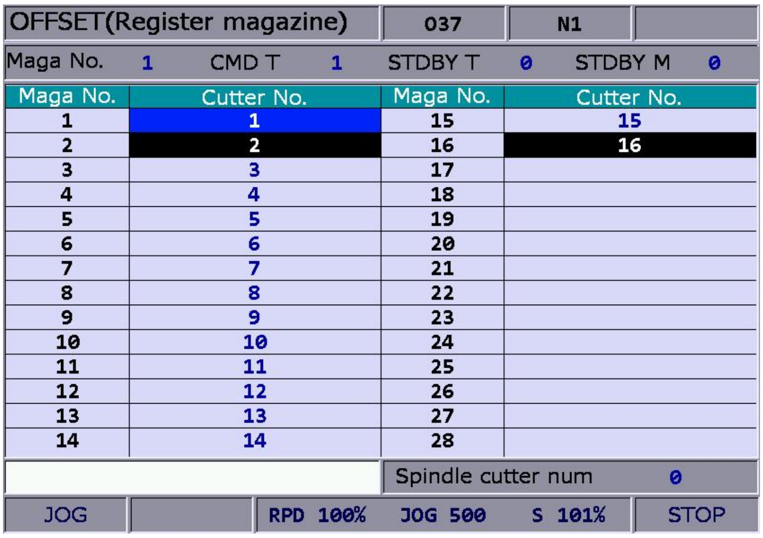
Spindle cutter num 0

JOG RPD 100% JOG 500 S 101% STOP

Figure 8.3.2

Example of tool magazine block:

Use this function to block a tool magazine adjacent to one that has a large diameter tool. Blocking these two tool magazines can stop the operation of an improper tool ID calling program and protects tools from colliding with large-diameter tools in neighboring tool magazine.



OFFSET(Register magazine)	037	N1	
Maga No. 1	CMD T 1	STDBY T 0	STDBY M 0
Maga No.	**Cutter No.**	**Maga No.**	**Cutter No.**
1	1	15	15
2	2	16	16
3	3	17	
4	4	18	
5	5	19	
6	6	20	
7	7	21	
8	8	22	
9	9	23	
10	10	24	
11	11	25	
12	12	26	
13	13	27	
14	14	28	
		Spindle cutter num 0	
JOG		RPD 100% JOG 500 S 101%	STOP

Figure 8.3.3

Assumption: T1 holds a large diameter tool and sided by T2 and T16. To prevent them from interfering with each other, block T2 and T16 with this function as shown in the Figure 8.3.3.

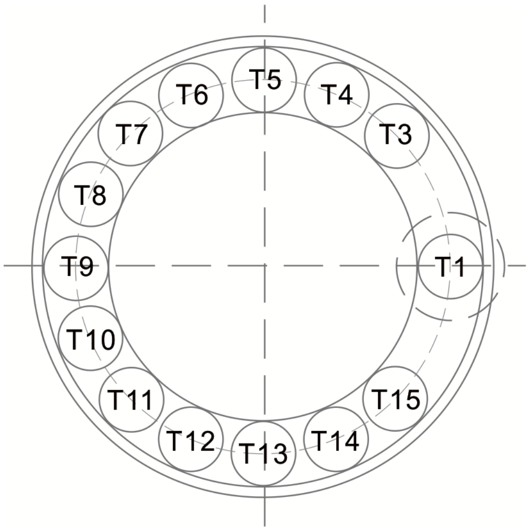


Figure 8.3.4

When T1 holds a large diameter tool, it may interference with adjacent tools as shown in Figure 8.3.4. Once blocked by this function, the T2 and T16 tool magazine position cannot be called anymore.

8

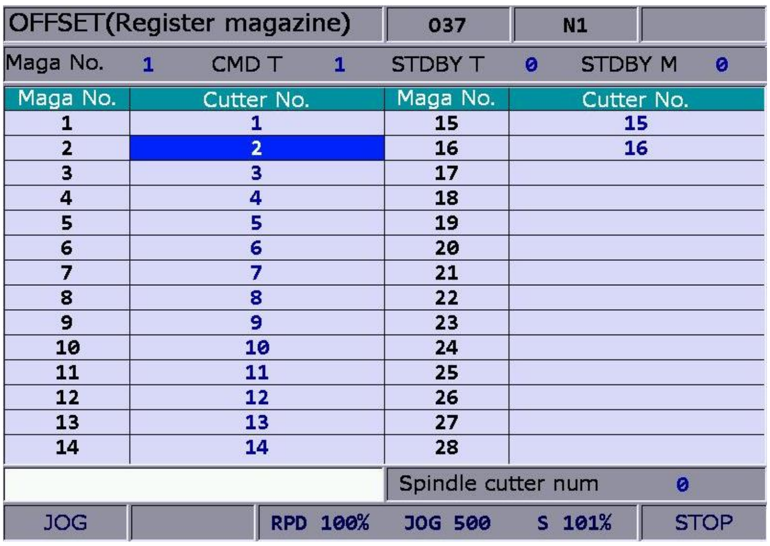
- **Tool magazine unlock:** This function can unlock the block tool magazine.

See the operation steps below:

1. Set the system to **Jog feeding mode**.
2. Press the **MAGA** to enter the screen of tool magazine data setup function.
3. Use , , , and keys to move the cursor to the assigned data fields.
4. Press the **UNLOCK** to unlock the tool magazine position.
5. Users also can directly enter the tool ID in the data field of locked tool magazine and then press the **ENTER** key to unlock the tool magazine position.

8.3.1 Multi-tool magazines management function

For applications that require multiple tool magazine management systems, with the permission, users may open the multi tool magazine management function through the tool magazine parameter. Users may assign a different number of tool pots for each tool magazine as well as the corresponding tool ID. The [Tool Magazine 1 and 2] function bars are used for managing tool ID in either tool magazine system. Please contact an equipment dealer/service provider for multi tool magazine relevant functions.


OFFSET(Register magazine) 037 N1

Maga No. 1 CMD T 1 STDBY T 0 STDBY M 0

Maga No.	Cutter No.	Maga No.	Cutter No.
1	1	15	15
2	2	16	16
3	3	17	
4	4	18	
5	5	19	
6	6	20	
7	7	21	
8	8	22	
9	9	23	
10	10	24	
11	11	25	
12	12	26	
13	13	27	
14	14	28	

Spindle cutter num 0

JOG RPD 100% JOG 500 S 101% STOP

Figure 8.3.1.1

Note:

1. The tool magazine ID can be set up in "**Jog feeding mode**" (JOG) only. The tool magazine setup option will not be shown in other modes.
2. The special user permission is a must before doing tool magazine ID setup or reset.
3. No tool IDs are identical in one tool magazine. When assigning one existing tool ID, the system will re-number this tool ID. In this case, the tool ID recorded at each address in tool magazine will be different to avoid incorrect tool calling.
4. When the spindle's initial tool ID is set to T0, once T0 is placed in one tool magazine, that tool magazine is recorded as the position of T0 and cannot be blocked. When the field of tool magazine is T0, it is not allowed to block and the message "**Tool ID Tool magazine cannot be blocked**" will pop up.

8.4 Macro variable

Use variable input setup of this function along with variable command for various MLC data I/O, condition computing, and controls. The macro variable function covers local, global, and retaining variables with values in double format.

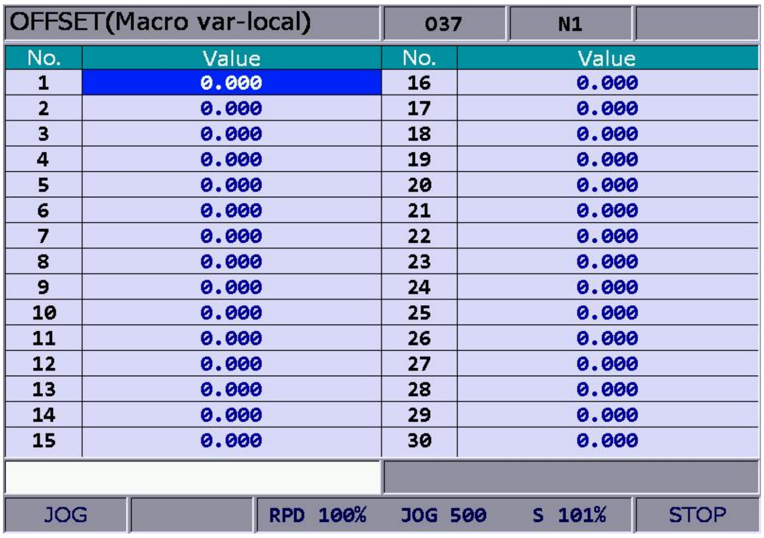


Figure 8.4.1

8.4.1 Local variable

Local variables are used by the macro program in the local area and are **numbered from 1 ~ 50.**

1. Press the **OFS** key to enter the screen of [Offset].
2. Press the **Macro** to enter the screen of variable entry.
3. Press the **Local** to enter the entry screen for field ID starting with number 1.
4. Use ↑, ↓, ←, and → key to move the cursor to the desired variable data field.
5. Enter variable value and press the **ENTER** key to complete the setting.

8.4.2 Global variable

Global variables are variables shared by the main programs, subroutines, and macro program. They are **numbered from 51 ~ 250.**

See the operation steps below:

1. Press the **OFS** key to enter the screen of [Offset].
2. Press the **Macro** to enter the screen of variable value entry.
3. Press the **Global** to enter the entry screen for field ID starting with number 51.
4. Use ↑, ↓, ←, and → key to move the cursor to the desired variable data field.
5. Enter variable value and press the **ENTER** key to complete the setting.

8.4.3 Retaining variable

These variables retain system data after power outage. They are **numbered from 1601 ~ 1800.** See the operation steps below:

1. Press the **OFS** key to enter the screen of [Offset].
2. Press the **Macro** to enter the screen for variable value entry.
3. Press the **Hold** to enter the entry screen for field ID started with number 1601.
4. Use ↑, ↓, ←, and → key to move the cursor to the desired variable data field.
5. Enter variable value and press the **ENTER** key to complete the setting.

8.4.4 Expanded variable

500 expanded variables are provided for the system. They are numbered from 10001 ~ 10500.

1. Press the **OFS** key to enter the screen of [Offset].
2. Press the **MACRO** to enter the screen for variable value entry.
3. Press the **EXPAND** and it will switch to the entry screen beginning with 10001.
4. Use ↑, ↓, ←, and → key to move the cursor to the desired variable data field. Enter variable values.
5. Press the **ENTER** to complete the setting.

Diagnosis (DGN) Group 9

DGN group provides machining information, user variables, system monitoring, and parameter import / export function to optimize the system.

The DGN group provides machining information, user variable, system monitoring, gain adjustment, and system interface functions to optimize the system. MLC diagnostics function is also provided for system to do MLC status monitoring or forced device ON/OFF, and password setup function for permission management. With this function, various system parameters can be imported / exported.

Note: Here we use **Framed Text** to indicate the keys in primary control panel. And **boldface letter** is used for indicating the function key.

9.1 Machining information (PROCESS)

In the screen of [PROCESS], users can set up the number of machining workpieces that have been completed (Completed stocks) and target machining workpieces (Target stocks). Users can also reset the time and count of machined pieces here.

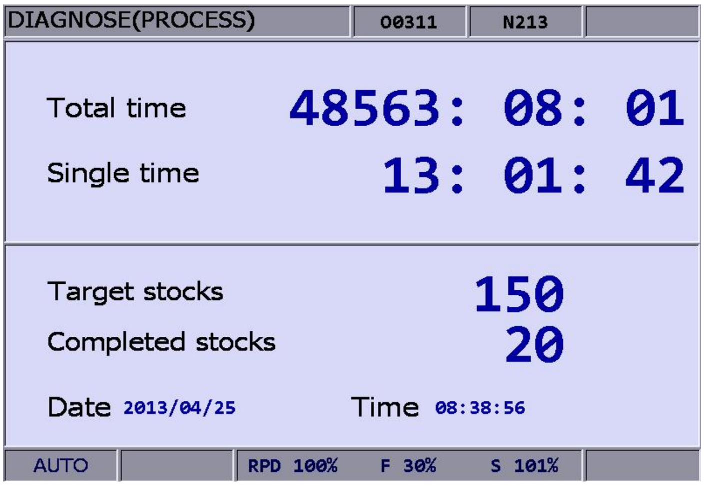


Figure 9.1.1

See the operation steps below for machining information setup:

1. Press the **DGN** key to enter the page of [DIAGNOSE] .
2. Press the **PROCESS** to enter the screen for machining information [PROCESS].
3. Press the **Set NR**, the machining count setup screen will pop up as shown in the figure below.

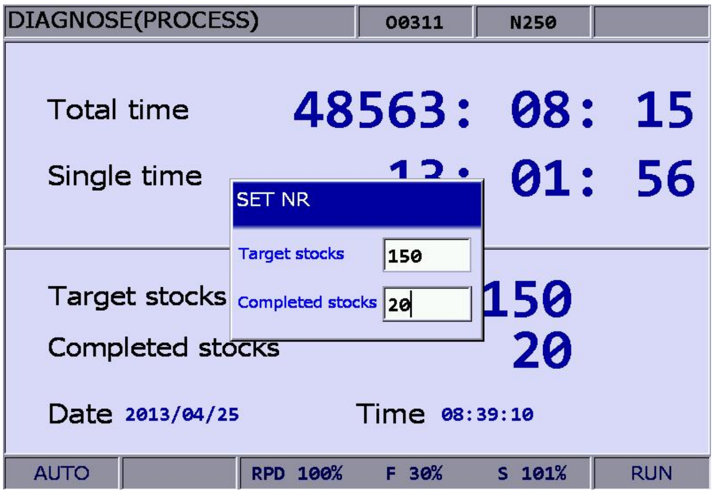


Figure 9.1.2

4. Use ↑ and ↓ keys to move the cursor to the specified field.
5. Enter the value in the range of 0 ~ 9999 and press the **ENTER** key to complete the setting.

In the screen of [PROCESS], users can reset the machining time (Total/Single) and completed stocks. See the operation steps below for clearing the machining time:

1. Press the **DGN** key to enter the screen of [DIAGNOSE]
2. Press the **PROCESS** to enter the setting page for machining information [PROCESS].
3. Press the **CLR TIME** and the confirmation dialog box will pop up.
4. Press **Y** (Yes) key and press the **ENTER** to reset the machining time of a single workpiece.

See the operation steps below for clearing the machining count:

1. Press the **DGN** key to enter the page of [DIAGNOSE].
2. Press the **PROCESS** to enter the page for machining information [PROCESS].
3. Press the **CLR NR** and the confirmation dialog box will pop up.
4. Press **Y** (Yes) key and press the **ENTER** to reset the count of workpieces that have been machined.

9.2 User variable

Function of user variable includes system variable, user variable and equipment variable. System variable is for monitoring the specific variable. User variable and equipment variable enable users to update and display device data in the embedded registers (range D512 ~ D1023). By displaying device types in registers users may change and monitor settings in the registers (D512 ~ D1023) for easier relevant devices control.

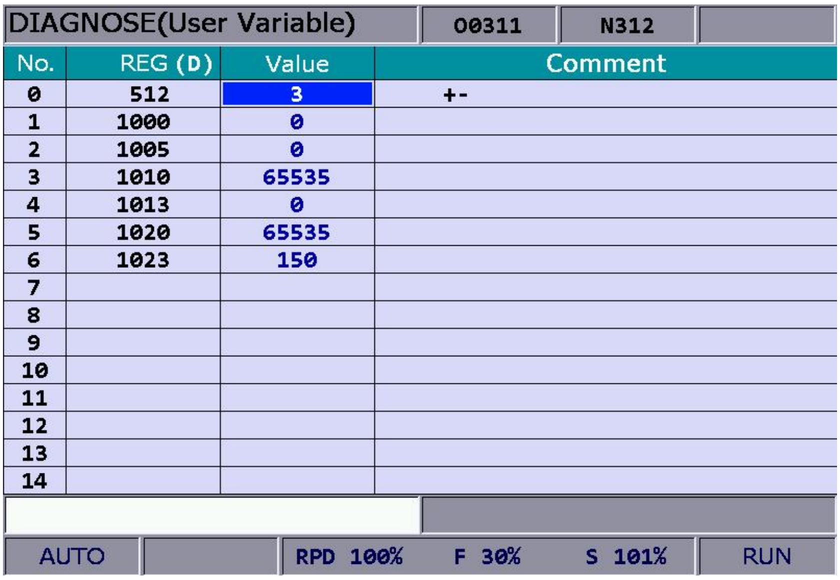

DIAGNOSE(User Variable)
O0311
N312
No.
REG (D)
Value
Comment
0 512 3 +-
1 1000 0
2 1005 0
3 1010 65535
4 1013 0
5 1020 65535
6 1023 150
7
8
9
10
11
12
13
14
AUTO
RPD 100% F 30% S 101%
RUN

Figure 9.2.1

See the operation steps below:

1. Press the DGN key to enter the page of [DIAGNOSE].
2. Press the **User VAR** to enter the setting screen.
3. Press the **USER VAR** to enter the setting screen. Or press the **M VAR** to enter the setting screen of equipment variable.
4. Use ⇧ and ⇩ or PAGE UP and PAGE DN keys to move the cursor to the specified data field.
5. Enter the desired register number (D512 ~ D1023). Then, press the ENTER key to display the register data.
6. Move the cursor to the specified register data field and enter the proper value. Then, press ENTER key to complete the setting for the register.
7. Use the **US DEC**, the **HEX**, the **S DEC** or the **FLOAT** to select the display format.
8. Move the cursor to the data field that requires to be deleted. Press the **DEL** to delete the data.

9.3 MLC

The MLC diagnostics function displays current status of each MLC device for monitoring and forced ON/OFF. This helps users in inspecting system status or driving MLC device and provides MLC editing function as shown in Figure 9.3.1. The MLC diagnostics function covers bit device status, register status, device monitoring, MLC status operation and MLC editing function. See the sections below for operation steps.

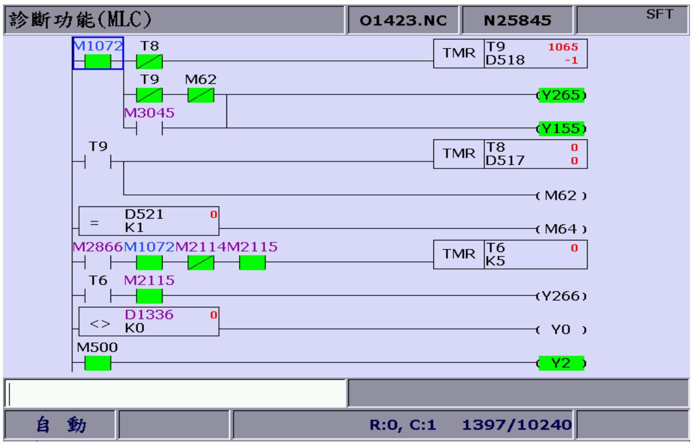


Figure 9.3.1

9

9.3.1 Bit

MLC programs apply many device commands to trigger ON/OFF operation. Status of these devices can be seen in this function screen. The bit function displays bit type device of MLC, searches devices, and forces ON/OFF operation. See the operation steps below: (illustrated with M device)

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **MLC** to enter the sub menu of MLC diagnostics.
3. Press the **Bit** to enter the screen of bit device status.
4. Press the **M** to switch to device M status display as shown in figure below.

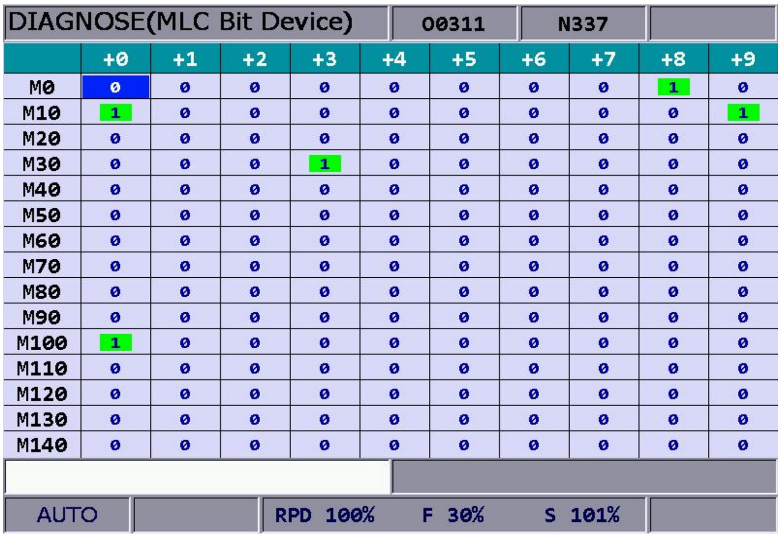


Figure 9.3.1.2

Move the cursor or search for the specified device field with the step 1 ~ 4. See step 5 for device searching.

5. Enter the device name (e.g. 107) and press the **M** to search the desired device (M107). The device status can be changed only when the system is in "NON-auto" mode. See step 6 for forced ON/OFF operation.
6. Select the device for the desired status change, press **1** and press the **ENTER** key to force it ON (if it is in OFF status) or press **0** and press the **ENTER** key to force it OFF (if it is in ON status).

9.3.2 Register

Please refer to **Section 9.3.1.** As most system functions are triggered by MLC program, the MLC device features both bit and word type devices. This section explains the operation on word type devices. See the operation steps below: (illustrated with register T)

1. Press the $\boxed{\textbf{DGN}}$ key to enter the screen of [DIAGNOSE].
2. Press the **MLC** to enter the sub menu of MLC diagnostics.
3. Press the **REG** to enter the screen of register device.

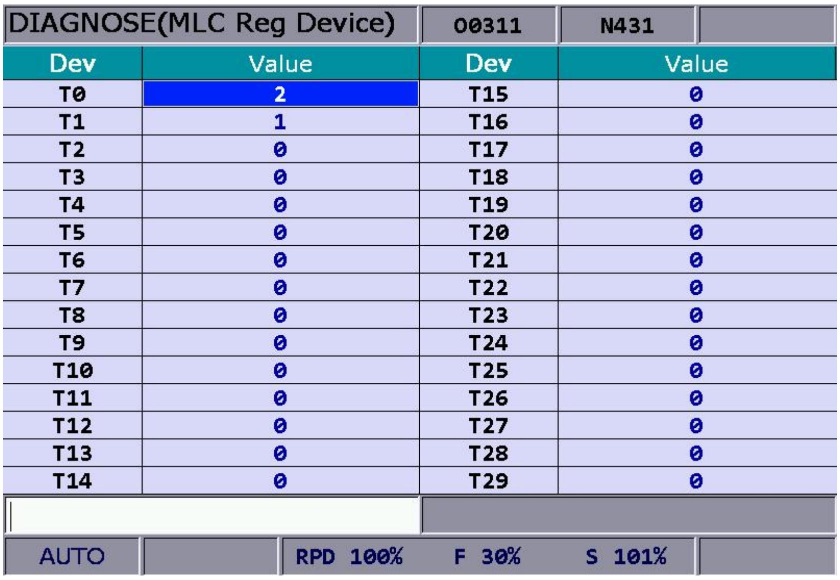


Figure 9.3.2.1

4. Press the **T** to enter the register T value setup page.
5. Enter the device name (e.g. "10") and press the **F1** key to search device T10.
6. Enter the setting value in the field and press the $\boxed{\textbf{ENTER}}$ key to complete the setting.
7. Switch to the function bar in the last page. Use **US DEC**, **HEX**, **S DEC** or **FLOAT** to select the display format.

9

9.3.3 Device monitoring

This function sets up monitoring functions for up to 45 devices.

See the operation steps below:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **MLC** to enter the sub menu of MLC diagnostics.
3. Press the **DEV MON** to enter the screen that displays device name as shown in Figure 9.3.3.1.

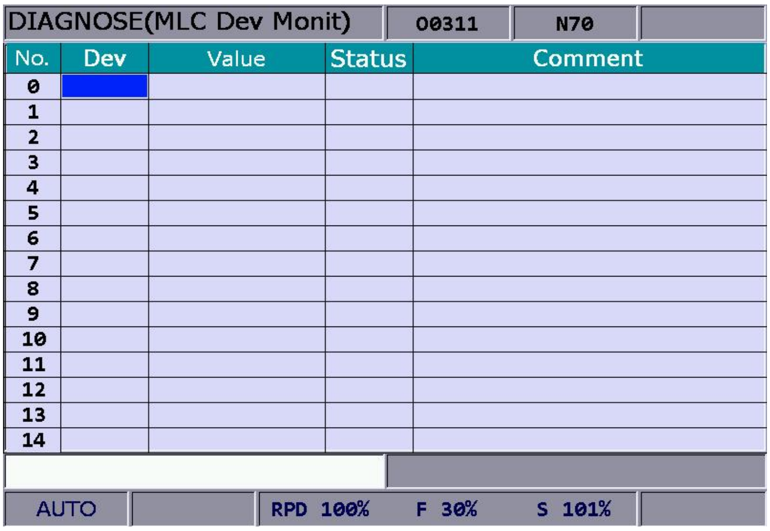


Figure 9.3.3.1

4. Enter the device name to be monitored as shown in figure 9.3.3.2. Up to 45 monitoring data entries can be set.

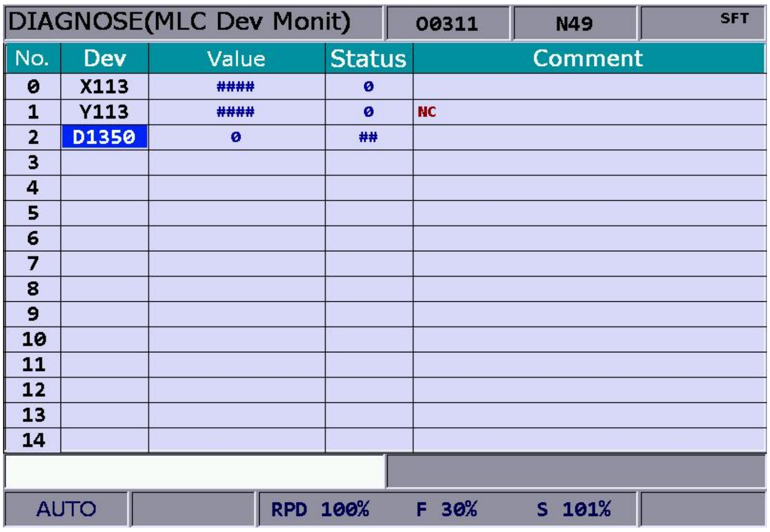


Figure 9.3.3.2

Device: Enter the name of device to be monitored in the highlighted field.

Value: Set up device status in the highlighted field.

Status: Enter digit 0 or 1 to set device status.

Different numeral systems can be used to switch between views of user settings including signed or unsigned decimal, hexadecimal numeral and floating point numerals. See Figure 9.3.3.3 for hexadecimal conversion and Figure 9.3.3.4 for floating point display.

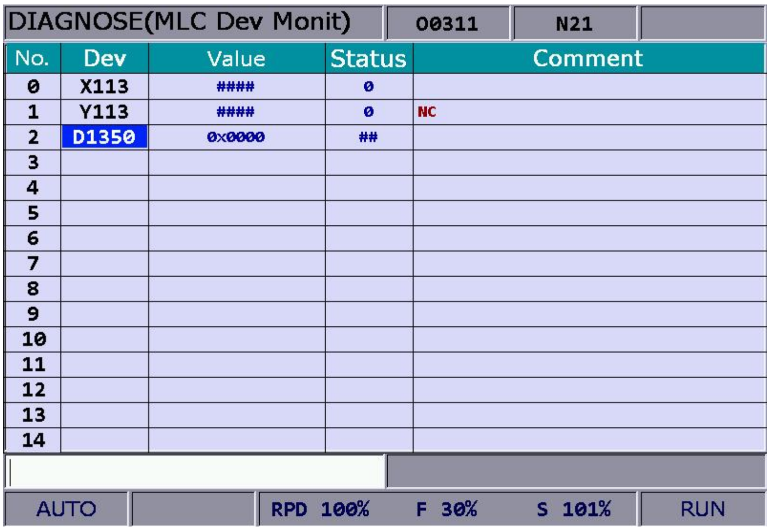


Figure 9.3.3.3

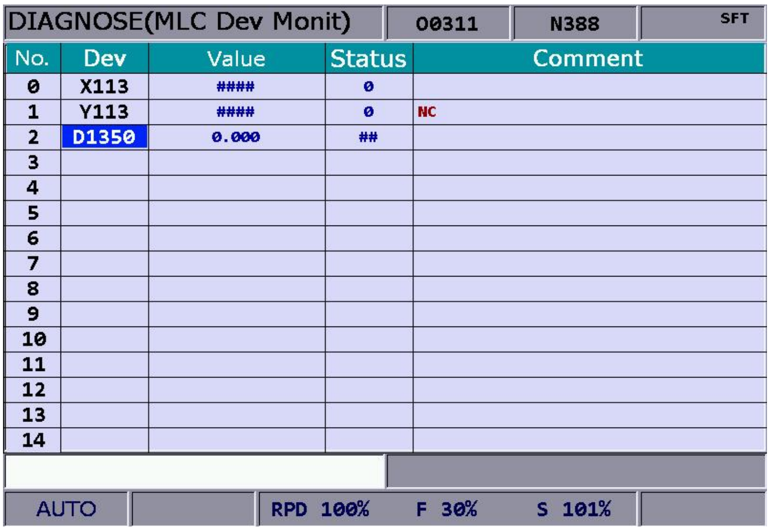


Figure 9.3.3.4

9.3.4 Search line

Most system functions rely on devices triggered by MLC programs which are basically a set of command lines. This function enables users to search a program by line number.

See the operation steps below for searching a desired line in a MLC program.

1. Press the $\boxed{\text{DGN}}$ key to enter the screen of [DIAGNOSE].
2. Press the **MLC** to enter the sub menu of MLC diagnostics.
3. Enter the specified MLC program line number and press the **JUMP TO** to go to the target line.

9.3.5 Editor

The edit function in DGN group can manage and edit MLC program. Its operation interface enables users to edit the MLC program directly. **This function can be run in "Edit mode" only.**

■ **Basic MLC command**

A basic MLC command (including: LD, LDI, LDP, LDF, OUT, APP, —, ｜, INV) can be created with the functions described in this section. See Figure 9.3.5.1 for illustration.

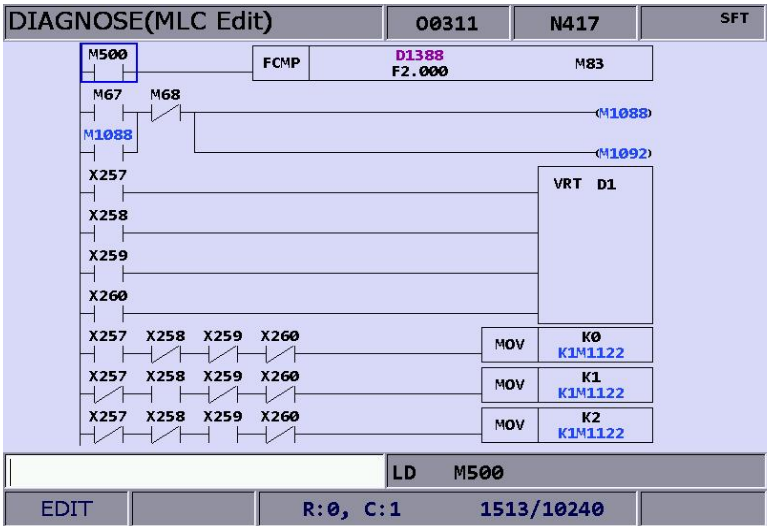


Figure 9.3.5.1

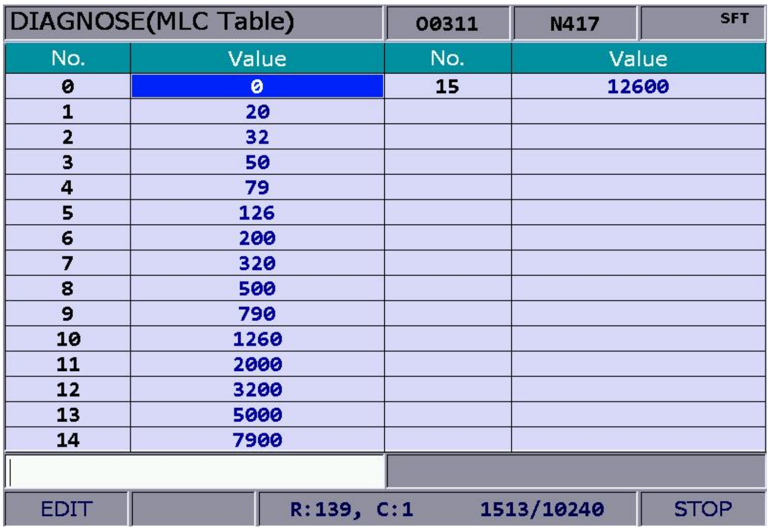
See the operation steps below for command LD:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **MLC** to enter the sub menu of MLC diagnostics.
3. Press the **EDITOR** to enter the MLC program editor screen as shown in Figure 9.3.5.1.
4. Use ↑, ↓, ←, and →keys to move the cursor to the specified edit place.
5. Enter the device name and press the **LD** and the device will be created successfully.

The steps described above apply to the creation of basic commands LDI, LDP, LDF, OUT, APP while step 1 ~ 4 apply to commands "—" and "｜". Then, use the corresponding function key to complete the command as described above.

The labeling function in MLC program is used to divide the section of the program and can be set in MLC program.

To assign values from MLC table, users can press the function key to enter the setting page which shown as the figure below.



DIAGNOSE(MLC Table)		O0311	N417
No.	Value	No.	Value
0	0	15	12600
1	20		
2	32		
3	50		
4	79		
5	126		
6	200		
7	320		
8	500		
9	790		
10	1260		
11	2000		
12	3200		
13	5000		
14	7900		

Figure 9.3.5.2

■ **Editing (cut, copy, and paste)**

This is an MLC exclusive editing function. Users can use it for single line delete, cut, or copy or do the same to the MLC device command by circling. After an MLC program is edited, load it for compiling and saving.

See the operation steps below for the MLC editing function:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **MLC** to enter the sub menu of MLC diagnostics.
3. Press the **EDITOR** to enter the MLC program editing page as shown in Figure 9.3.5.1.
4. Use ↑, ↓, ←, and → keys to move the cursor to the desired edit place.
5. Repeatedly press the ▶ key to move the cursor to the last row of the function page in this layer.
6. Select the corresponded function key, such as **CUT** to edit the selected line as required.

For MLC program editing, please press the relevant function key. Available functions are: circle, delete, cut, copy, paste, insert and delete line.

- **Symbol**

This function enables users to search, delete, copy, and paste various types of devices. Available MLC program devices are represented by symbols: X, Y, M, A, T, C, D, P and I.

See the operation steps below:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **MLC** to enter the sub menu of MLC diagnostics.
3. Press the **EDITOR** to enter the MLC program editing screen as shown in Figure 8-3-6.
4. Use ↑, ↓, ←, and → keys to move the cursor to the desired edit place.
5. Press the ▶ key to switch the function bar to the third row of the function page in this layer.
6. Press the **SYMBOL** to enter the device symbol function bar display.
7. Select the device type specific function key (e.g. Device X). Press the **X** to enter X device specific list and do delete, copy or paste function as desired.

The same operation steps (Section III: Symbol) apply to other symbols.

- **MLC load, import, and export**

After a MLC program is edited, it is required to save it for re-compiling. The saving function includes compiling and saving the file. Then, users should re-start the system to update the MLC program. Import and export MLC files can be done by using the corresponding function key.

9.3.6 Operation

The MLC program starts running automatically after the system is power on. The operation function can be used to manually switch the status of MLC program. That is, users can manually switch MLC running status from ON to OFF and vice versa. This is usually used for testing or inspecting system's MLC devices.

See the operation steps below:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **MLC** to enter the sub menu of MLC diagnostics.
3. Press the **SET** to enter the MLC execution status screen.
4. Press the **RUN/STOP** to switch execution status of MLC program.

Note: The status information can be viewed when "MLC stops" after the MLC program execution is halted.

The function option can be used to force ON or OFF a MLC device.

- Operation steps for forced ON:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **MLC** to enter the sub menu of MLC diagnostics.
3. Press the **SET** to enter the MLC execution status screen.

4. Use , , , and keys to move the cursor to the specified device position.
5. Press the **ON** to switch on the device.

■ Operation steps for forced OFF:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **MLC** to enter the sub menu of MLC diagnostics.
3. Press the **SET** to enter the MLC execution status screen.
4. Use , , , and keys to move the cursor to the specified device position.
5. Press the **OFF** to switch off the device.

9.4 System monitoring

Computing results of the system can be displayed by type with this function, providing real data for users.

9.4.1 Servo monitoring

This function enables users to monitor the status of the servo drive including channel ports of each axis and the servo status. It provides the current status of the servo drive connected to the system as shown in Figure 9.4.1.1. In the figure below, the servo status of axis Z and the spindle are both Off and axis X and Y remain ON.

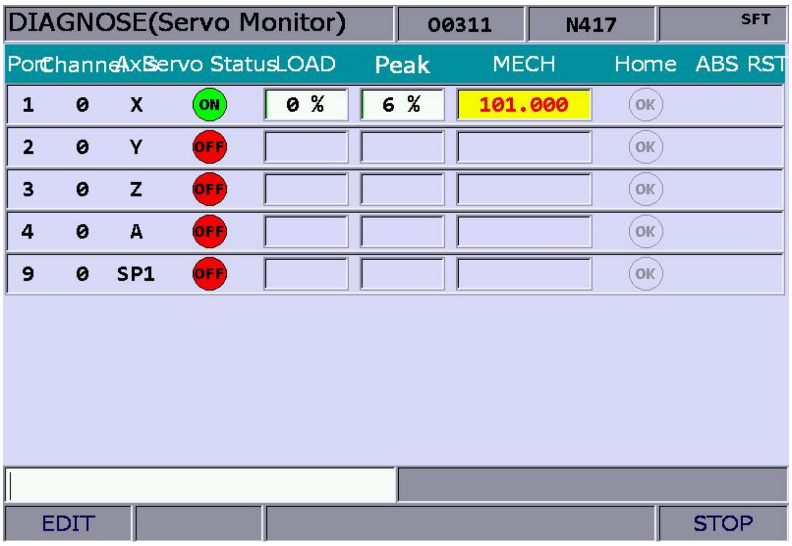


Figure 9.4.1.1

See the operation steps below:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **SYS MONI** to switch to the system monitoring screen.
3. Press the **SERVO** to enter the servo monitoring screen.

9.4.2 I/O monitoring

NC system can add an external control switch through its I/O expansion module. Users can monitor the status of the expansion control panel connected to the I/O port.

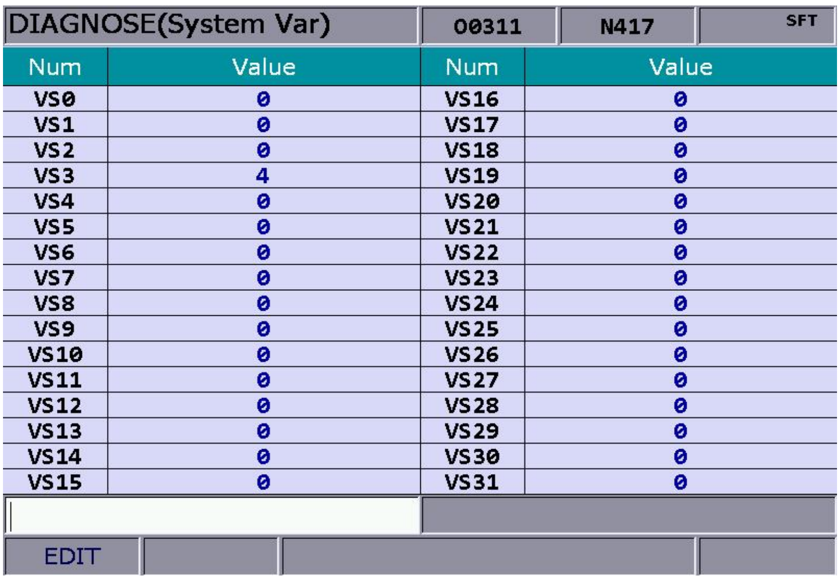
See the operation steps below:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **SYS MONI** to enter the screen for system monitoring.
3. Press the **I/O MONI** to enter the status monitoring screen for the I/O expansion module.

9.4.3 Variable monitoring

■ System variables: VS0 ~ VS31 and VS100 ~ VS131. See the operation steps below:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **SYS MONI** to enter the system monitoring function screen.
3. Press the ▶ key to move the cursor to the second row of the function page in this layer.
4. Press the **VAR MONI** to display the variable monitoring screen.
5. Press the **SYS VAR** to enter the system variable monitoring screen.
6. Use **PAGE UP** and **PAGE DN** keys to scroll to the screen containing the desired variable.
7. Or you can enter the full name of the specified system variable and press the **ENTER** key or enter the variable code and press the **SYS VAR** to search the desired system variable screen.



DIAGNOSE(System Var) | 00311 | N417 | SFT

Num	Value	Num	Value
VS0	0	VS16	0
VS1	0	VS17	0
VS2	0	VS18	0
VS3	4	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

EDIT

Figure 9.4.3.1

- **Channel variable**: VC0 ~ VC31, VC100 ~ VC131 and VC200 ~ VC231.

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **SYS MONI** to enter the system monitoring function screen.
3. Press the ► key to move the cursor to the second row of the function page in this layer.
4. Press the **VAR MONI** to enter the variable monitoring screen.
5. Press the **CH VAR** to enter the channel monitoring screen.
6. Use PAGE UP and PAGE DN keys to scroll the screen containing desired variable.
7. Or you can enter the full name of the specified system variable and press the ENTER key or enter the variable code and press the **CH VAR** to search the desired channel monitoring screen.

- **Axis variable**: VA0 ~ VA31, VA100 ~ VA131 and VA200 ~ VA231.

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **SYS MONI** to enter the system monitoring function screen.
3. Press the ► key to move the cursor to the second row of the function page in this layer.
4. Press the **VAR MONI** to enter the variable monitoring screen.
5. Press the **AXIS VAR** to enter the axis variable monitoring screen.
6. Use PAGE UP and PAGE DN keys to scroll the screen containing the desired variable.
7. Or you can enter the full name of the specified system variable and press the ENTER key or enter the variable code and press the **AXIS VAR** to search the desired axis variable screen.

- **Interface variable:** VH0 ~ VH31, VH200 ~ VH231 and VH400 ~ VH431 and VH800 ~ VH863.

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **SYS MONI** to enter the system monitoring function screen.
3. Press the ► key to move the cursor to the second row of the function page in this layer.
4. Press the **VAR MONI** to enter the variable monitoring screen.
5. Press the **IF VAR** to enter the interface variable monitoring screen.
6. Use PAGE UP and PAGE DN keys to scroll the screen containing the desired variable.
7. Or you can enter the full name of the specified system variable and press the ENTER key or enter the variable code and press the **IF VAR** to search the desired interface variable screen.

- **MLC variable**: VM0 ~ VM49

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **SYS MONI** to enter the system monitoring function screen.
3. Press the ► key to move the cursor to the second row of the function page in this layer.
4. Press the **VAR MONI** to enter the variable monitoring screen.

5. Press the **MLC VAR** to enter the MLC variable monitoring screen.
6. Use PAGE UP and PAGE DN keys to scroll to the screen containing the desired variable.
7. Or you can enter the full name of the specified axis variable and press the ENTER key. Or enter the variable code and press the **MLC VAR** to search the desired variable screen.

9.5 Password setting

This function enables users to set up different permission levels for the system (system maintenance), equipment (mechanical equipment) and users (operation). It prevents unauthorized users from changing system settings.

9.5.1 User permission

Users can set up User permission 1 and User permission 2. The permission function covers password change (PWD CHG), user account lockup (LOCK) and user account unlocking (UNLOCK). The password is composed of up to four alphanumeric characters (symbols excluded). See the operation steps below for permission lock and unlock:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to enter the password setup function bar.
3. Press the **S SCP** to enter the system permission lock/unlock function bar.
4. If the system permission is unlocked, press the **LOCK** to lock system permission.
5. If the system permission is locked, press the **UNLOCK** and an entry dialog box will pop up for users to enter permission password.
6. Enter a valid password and press the ENTER key to unlock the permission.

Steps to inspect the system:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to display the password setup function bar.
3. Press the **S SCP** to enter the system permission lock/unlock function bar.
4. When the system permission is unlocked, press the **SYS CHECK** to see if there is any error occurs. If the item is checked, an error is found under the checked item.

9.5.2 Equipment permission

This function covers password change, permission lock, permission unlock, user 1 reset and user 2 reset. The password is composed of up to four alphanumeric characters (symbols excluded).

See the operation steps below for changing equipment permissions:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to enter the password setup function bar.
3. Press the **M SCP** to enter the equipment permission function bar.
4. Press the **PWD CHG** and an entry dialog box will pop up as shown in Figure 9.5.2.1. Enter old password, new password, and new password again (for confirmation) as prompted.
5. Enter the passwords as prompted and press the ENTER key.

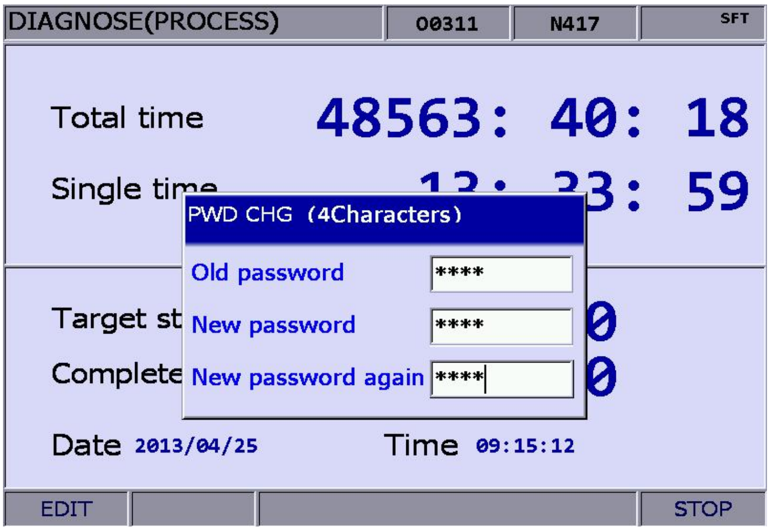


Figure 9.5.2.1

See the operation steps below for equipment permission resetting:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to enter the password setup function bar.
3. Press the **M SCP** to enter the equipment permission function bar.
4. Press the **UNLOCK** and an entry dialog box will pop up for users to enter the password when equipment permission is locked.
5. Enter a valid password and press the ENTER key to revoke the equipment permission.

Note: The default password of equipment permission is 0000, which means the permission is unlocked and all functions can be accessed. When the password is changed, the equipment permission is enabled and it means the related functions can be accessed with the permission only.

See the operation steps below for equipment permission lock up:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to enter the password setup function bar.
3. Press the **M SCP** to enter the equipment permission function bar.

4. Press the **LOCK** to lock permission when it is revoked.

User reset function allows the equipment supplier to reset the user's password. Once the client forgets the password, the equipment supplier is able to reset as the default password. This function is active only when the password is not the default value. See below for the operation steps.

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to display the password setup function bar.
3. Press the **M SCP** to display the equipment permission function bar.
4. Press **RST U1** / **RST U2** to reset the user's password.

Function Enabled is to enable/disable the group function. Once the group is canceled, the function will be disabled after the system is re-started up. See below for the operation steps.

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to display the password setup function bar.
3. Press the **M SCP** to display the equipment permission function bar.
4. Press the **ENABLE** to enter the setting screen to enable the group function.
5. Use ↑ and ↓ keys to move the cursor to the field to be cancelled. Press the **ENTER** key to cancel the selection. Then, press the **F1** key once the setting is done. The setting will take effect after the system is re-started up.
6. If you wish to cancel the group function, press the **CANCEL** to exit the the setting page of group function.
7. If you wish to restore the system to the default setting, press the **DEFAULT** to restore the system to the initial setting of group function.

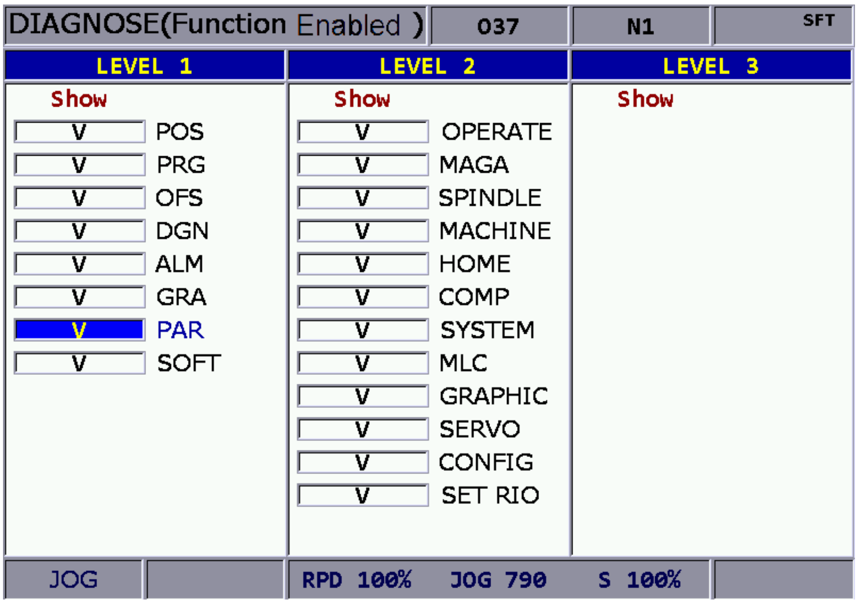


Figure 9.5.2.2

Restore Function

When the numerical control system has any error or the system data is seriously damaged, this function enables users to restore the damaged data through system backup. Users need to enter the restore screen to select the item to be restored. Please note that permission is required to apply this function.

See below for the operation steps:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to display the password setup function bar.
3. Press the **M SCP** to display the equipment permission function bar.
4. Press the **RESTORE** and enter the screen to select the item to be restored. Use ↑, ↓, ←, and → keys to move the cursor. Then, press the ENTER key to select the item to be restored.
5. To cancel the selection: Press the ENTER key on the checked iem to cancel the selection.
6. Press the **OK** to restore the system.

9.5.3 User permission

Users can set up User permission 1 and User permission 2. The permission function covers password change (PWD CHG), user account lockup (LOCK) and user account unlocking (UNLOCK). The password is composed of up to four alphanumeric characters (symbols excluded).

See the operation steps described below for changing user password (illustrated with User permission 1):

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to enter the password setup function bar.
3. If [User permission 1] is locked, press the **U1 SCP** and the password entry dialog box for unlocking [User permission 1] will pop up.
4. Enter valid password and press the ENTER key to to unlock [User permission 1] and display relevant function items.
5. Press the **PWD CHG** and an entry dialog box pops up for users to enter old password once and new password twice for confirmation.
6. Enter passwords as prompted and press the ENTER key to complete the setting.

See the operation steps described below for user permission lockup:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to display the password setup function bar.
3. When [User permission 1] is unlocked, press the **U1 SCP** to display relevant function items.
4. Press the **LOCK** to lock [User permission 1].

Note: The function of user permission is the same as equipment permission. Its default password is 0000, which means all functions are available. If the user password is changed, the user permission is enabled.

9.5.4 Timed use

Users can assign a timed use of the controller to limit its use in a set period of time. The system controls given days or hours for the use of the controller automatically. Users can unlock or reset the time limit only with valid permission when the time limit is active. The "deadline" in the time limit screen remains blank when no time limit is set or the limit is unlocked as shown in Figure 9.5.4.1. If there is an active time limit in existence and it is locked, the "**deadline**" in the time limit screen indicates a valid due date as shown in Figure 9.5.4.2:

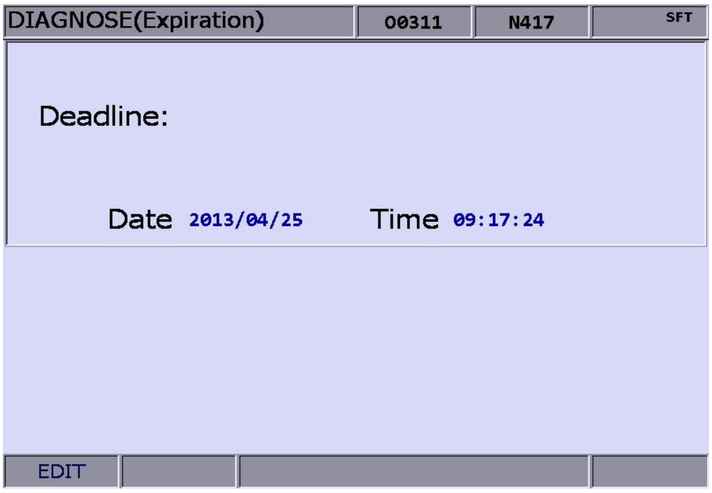


Figure 9.5.4.1

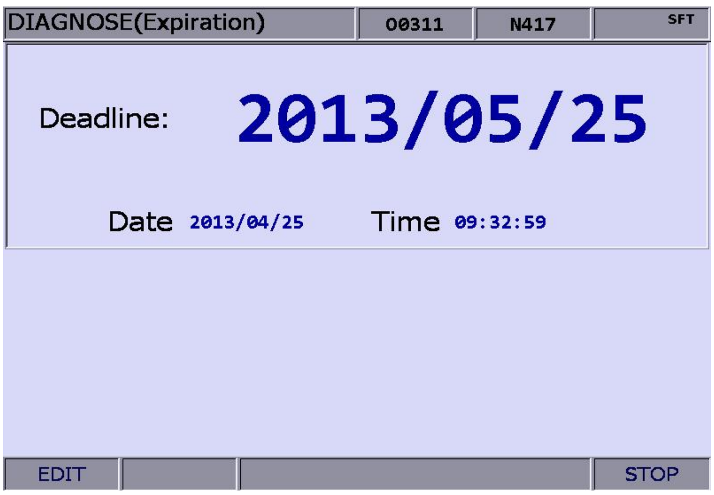


Figure 9.5.4.2

This screen helps users to know to which date the controller can be used normally. After the due date, the system will be locked unless the time limit is unlocked or extended to a later date. Otherwise, no G code program can be executed manually or automatically. Please contact the dealer/service provider in case it is overdue.

This function enables users to set up a time limit when there is no time limit in existence. See the operation steps described below for **time limit** setup:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to enter the password setup function bar.
3. Press the **EXPIRE** to display the time limit information.
4. Press the **SETTING** to enter the limit setup page.
5. After entering the password of legal permission, the system's time limit control is activated.

See the operation steps described below for revoking a time limit. Please contact the dealer/service provider for further information.

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to enter the password setup function bar.
3. Press the **EXPIRE** to display the remaining time information.
4. Press the **RELEASE** and a dialog box will pop up that requires users to enter the start code. See Figure 9.5.4.3

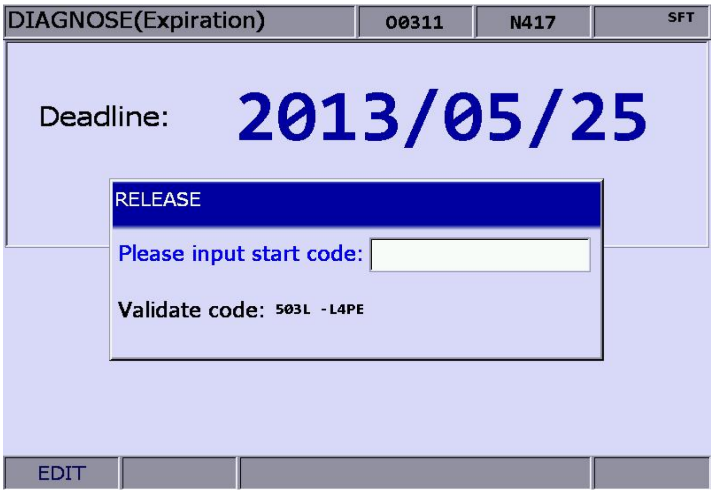


Figure 9.5.4.3

5. With proper authorization, enter the start code and press the **ENTER** key. Then, restart the system, the time limit is now unlocked.

Note: After the time limit is unlocked, the "deadline" field turns blank, as shown in Figure 9.5.4.4. The screen indicates that the system does not have a time limit set up.

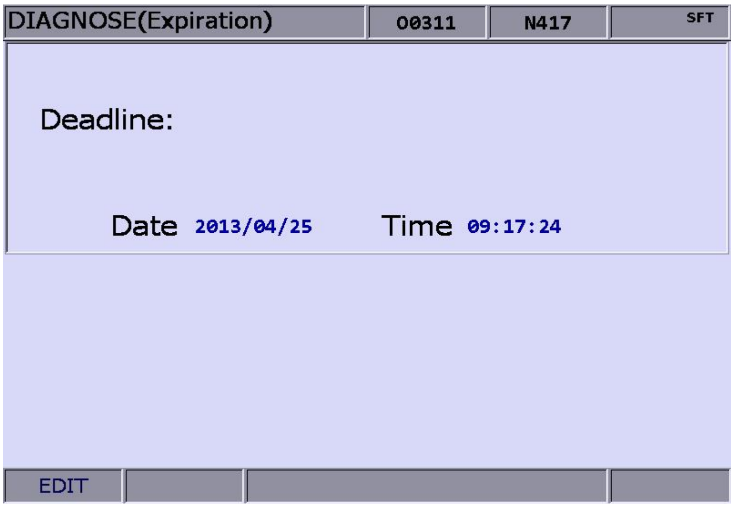


Figure 9.5.4.4

The management of time limit permission must go through the proper authorization to lock or unlock the time limit permission. When the time limit is activated, only when entering the correct password can the permission be unlocked. After the permission is unlocked, all time limit function is available, including password change and permission lock/unlock. The password is composed of up to 4 alphanumeric characters (symbol excluded).

See the operation steps described below for changing the password of time limit.

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to enter the password setup function bar.
3. Press the **EXPIRE** to display the remaining time information.
4. When the permission is locked, press the **EXP SCP** and the password entry dialog box will pop up for unlocking time limit.
5. Enter the valid password for time limit permission and press the **ENTER** key to unlock time limit permission and display relevant function items.
6. Press the **PWD CHG**, and an entry dialog box will pop up for users to enter old password once and new password twice for confirmation.
7. Enter passwords as prompted and press the **ENTER** key.

See the operation steps described below for revoking the time limit permission.

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to enter the password setup function bar.
3. Press the **EXPIRE** to display the remaining time information.
4. When the permission is locked, press the **EXP SCP** and the password entry dialog box will pop up for unlocking time limit permission.
5. Enter the valid password for time limit permission. Then, press the **ENTER** key to unlock the time limit permission and display relevant function items.

See the operation steps described below for locking the time limit permission.

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **PWD** to enter the password setup function bar.
3. Press the **EXPIRE** to display the remaining time information.
4. When the permission is unlocked, press the **EXP SCP** for displaying the relevant function items.
5. Press the **LOCK** and resume the permission lock.

9.6 System information

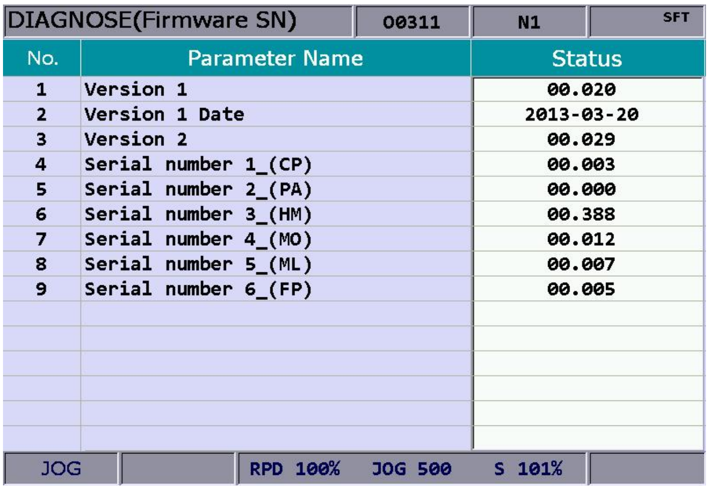
This function provides hardware and firmware program version of this system for system maintenance and performance optimization. It covers the options of system status, hardware and firmware serial number and equipment information.

See the operation steps described below for system status display:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **STATUS** to enter the screen of system information.
3. Press the **SYSTEM** to enter the screen of system status information.

The firmware serial number function displays the firmware version number as well. See the operation steps described below:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **STATUS** to enter the screen of system information.
3. Press the **FW SN** to display firmware version information as shown in the figure below.


DIAGNOSE(Firmware SN) | O0311 | N1 | SFT

No.	Parameter Name	Status
1	Version 1	00.020
2	Version 1 Date	2013-03-20
3	Version 2	00.029
4	Serial number 1_(CP)	00.003
5	Serial number 2_(PA)	00.000
6	Serial number 3_(HM)	00.388
7	Serial number 4_(MO)	00.012
8	Serial number 5_(ML)	00.007
9	Serial number 6_(FP)	00.005

JOG | RPD 100% JOG 500 S 101%

Figure 9.6.1

See the operation steps described below for hardware serial number display:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **STATUS** to enter the screen of system information.
3. Press the **HW SN** to display hardware version information.

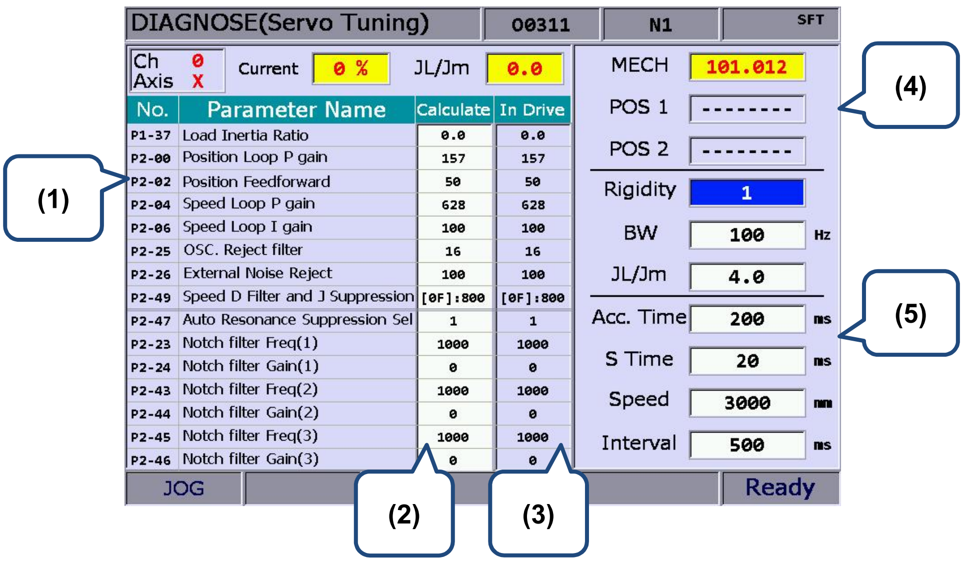
See the operation steps described below for equipment information display:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the **STATUS** to enter the screen of system information.
3. Press the **M STATUS** to display the equipment information.
4. Users can enter the equipment information in this page or press the **DEL** to delete the equipment information where the cursor indicates.

9.7 Gain adjustment [Tuning]

The auto gain adjustment enables the system and the servo drive to work out even better motion control to meet different mechanical requirements of various machines.
The NC controller accesses initial parameters of the servo and calibrates motion control with gain adjustment function. Then, it will send the result to the servo drive for unifying the control parameters of the controller and the servo drive. This brings the convenient when adjusting the gain and enhances the control accuracy for the system.

Sub menu items of this function are described with the function screen as shown in the figure below



(1) Servo parameter ID: Servo parameter Id and name;

(2) Calculated results after adjustments: Display calculated results after auto gain;

(3) Existing settings of the system: Indicates servo settings currently used by the system;

(4) Anchor point setup: Anchor point 1 / Anchor point 2;

(5) Adjustment conditions.

9

■ **Next axis:** This function switches axial gain settings. The auto gain can be adjusted by individual axis. After the first axis is adjusted, users need to switch to the next one for its adjustment. See the operation steps described below:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the ▶ key to enter the next function page.
3. Press the **TUNING** to enter the auto gain setup page.
4. Press the **NEX AX** to switch to the next axis for its axial gain parameters setup.

■ **Read the Servo:** After the auto gain adjustment function is activated, its parameter values have been synchronized with those of the servo. To accommodate the function of gain adjustment, the calculated results after auto gain operation are not written back to the servo. This function can be used to restore servo parameters. See the operation steps described below:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the ▶ key to switch to the next function page.
3. Press the **TUNING** to enter the auto gain setup page.
4. Press the **READ** to access parameter values from the servo.

■ **Start, Jog←, Jog→, Positioning 1, Positioning 2:** This sets up the operation of auto gain adjustment. It starts auto gain adjustment and sets up the positioning direction and operation. See the operation steps described below for continuous operation (Single-axis operation):

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the ▶ key to switch to the next function page.
3. Press the **TUNING** to enter the auto gain setup page.
4. Press the ▶ key to switch to the next function page.
5. Press the **JOG** ← to move to the left positioning point.
6. Press the **POS 1** to set the left positioning point.
7. Press the **JOG** → to move to the right positioning point.
8. Press the **POS 2** to set the right positioning point. The movement is now limited between both points.
9. Press the **RUN** to execute gain adjustment.
10. Press the **STOP** during auto adjustment. Then, the system will automatically calculate the best gain value.

■ **Gain calculation:** Users can change low-frequency rigid, bandwidth, or inertia ratio to fit individual machines. These values can be generated by this function automatically. See the operation steps described below for single axis operation:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the ▶ key to switch to the next function page.
3. Press the **TUNING** to enter the auto gain setup page.
4. Use ↑ and ↓ keys to move the cursor to the low-frequency rigid, bandwidth, or inertia ratio fields for entering settings respectively.
5. Press the **COMPUTE** to generate new gain values.

■ **Gain and resonance write-in:** New gain values are generated after the auto gain adjustment has stopped. If they are the expected optimization values, please use this function to write them in the servo drive. See the operation steps described below:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the ▶ function key to switch to the next function page.
3. Press the **TUNING** to enter the auto gain setup page.
4. Gain values are generated automatically after gain adjustment is complete.
5. Press the **WR GAIN** to write gain values in the servo drive. Then, press the **WR NOTH** to write resonance suppression values in the servo drive.

Note:

1. The newly generated gain adjustment results must be written in the servo drive before it can take effect.
2. After gain and resonance write-in function is executed, the servo parameters are updated and the old ones cannot be recovered. Please do the write-in with care.

■ **Tapping adjustment:** This function fine tunes the machine and servo for tapping application. See the operation steps described below:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the ▶ key to switch to the next function page.
3. Press the **TUNING** to enter the auto gain setup page.
4. Complete X-, Y- and Z-axis and spindle's gain adjustment first.
5. Repeatedly press the ▶ key to enter the last page.
6. Press the **TAP RIV** to switch to the operation screen.
7. Press the **TAP SET** again and the confirmation dialog box will pop up. Press Y (Yes) key and press the ENTER key to fine tune the machine for tapping.

■ **Servo parameter:** This function sets up the parameter for servo parameter display and setup in the gain adjustment screen:

1. Press the DGN key to enter the screen of [DIAGNOSE].
2. Press the ▶ key to switch to the next function page.
3. Press the **TUNING** to enter the auto gain setup page.
4. Repeatedly press the ▶ key to enter the last page.
5. Press the **SERVO** to enter the servo parameter screen.
6. Move the cursor to the specified field and type in relevant data. Then, press the ENTER key to set up a given field.

■ **Synchronized control:** This function can be applied when users need to synchronously control the master axis and slave axis. Before enabling this function, users have to complete the setting of parameters and channel. See the operation steps described below:

1. Firstly, complete the setting of parameter 361 ~ 366 and channel axis.
2. Press the DGN key to enter the screen of [DIAGNOSE].
3. Press the ▶ key to switch to the next function page.
4. Press the **TUNING** to enter the auto gain setup page.
5. Repeatedly press the ▶ key to enter the last page.
6. Press the **SYN** to switch to the operation screen.
7. Then, press the **POS SET** to complete the setting.

9

9.8 Import

The system features a parameter import/export function for managing system parameters. Users can import correct parameters recover the system and export the modified parameter files for backup. This function can only be used with proper permissions. It can efficiently troubleshoot the system with parameter errors.

See the operation steps described below for parameter import:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the ▶ key to switch to the next function page.
3. Press the **IMPORT** to display window of [FILE] (see Figure 9.8.1). Use ↑ and ↓ keys to select the file path and press the **ENTER** key to access the file.

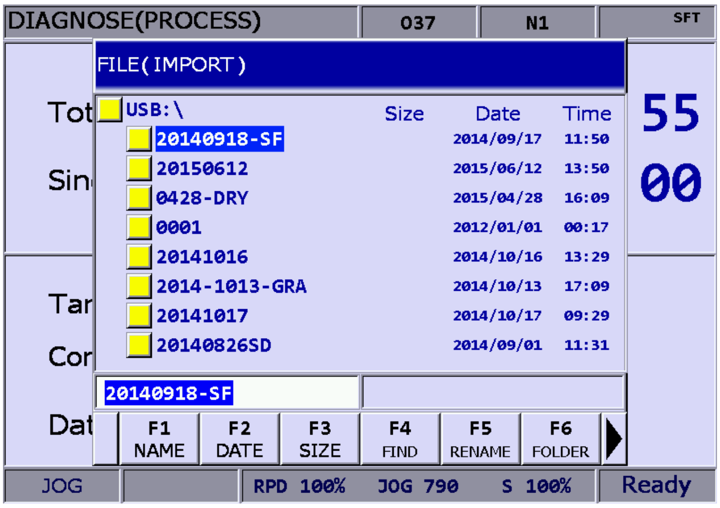


Figure 9.8.1

4. Use ↑, ↓, ←, and → keys to move the cursor. Press the **ENTER** key to check the parameter items to be imported.

5. Press the **SEL ALL** to select all or press the **CLR ALL** to cancel the selected items to be imported.

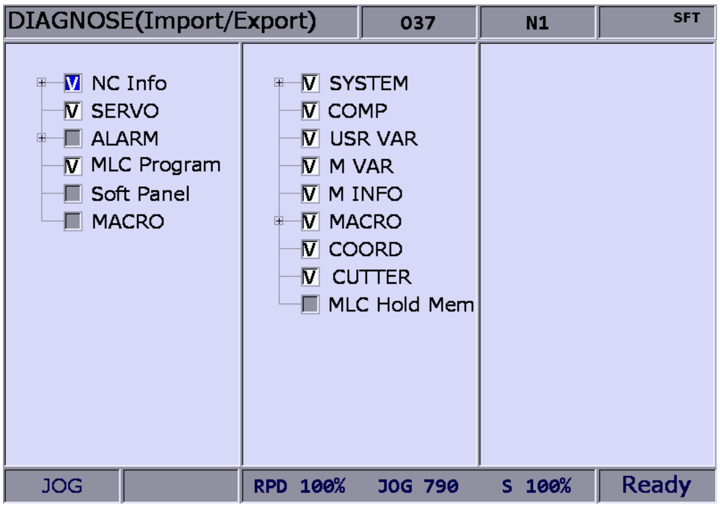


Figure 9.8.2

6. Press the **IMPORT** to display the dialog box for confirmation. Enter $\boxed{\text{Y}}$ (yes) and press the $\boxed{\text{ENTER}}$ key, the data in the file will be imported to the system. Then, the importing progress will be displayed until it is complete.

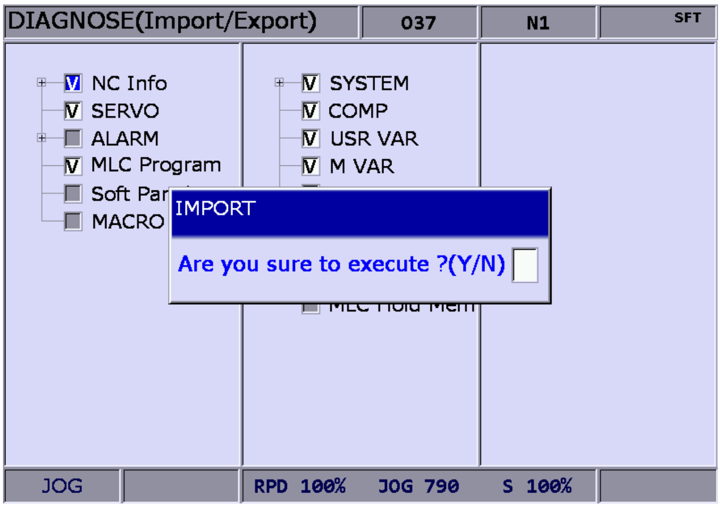


Figure 9.8.1

9.9 Export

Export function: System parameters may be modified to meet the requirements of different applications. After the system is optimized, this function can be used to export parameter values for backup and control. The exported file type includes parameter file, MLC and software panel. Please note that this function can be used only with proper permissions.

Type	Filename	Descriptions
Parameter file	PAR.ncp	NC information, servo parameters and alarms
MLC	MLC.gmc	MLC code
MLC	MLC.lad	Image file of MLC Ladder
MLC	MLC.lcm	Comments of MLC Ladder
Software panel	HMI.cin	Screen information and element property of the software panel
Software panel	HMI.img	Image file of software panel
Software panel	HMI.sci	Project of software panel

See the operation steps described below for parameter export:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Press the ▶ key to switch to the next function page.
3. Press the **EXPORT** to enter the screen of parameter export selection.
4. Use ↑, ↓, ←, and → keys to move the cursor. Press the **ENTER** key to check the items to export. Users may press the **SEL ALL** to select all or press the **CLR ALL** to cancel all the sleceted items to export.

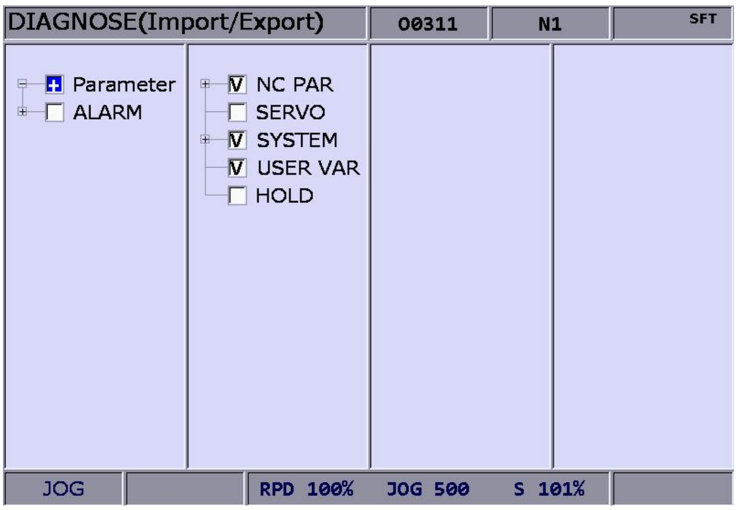


Figure 9.9.1

5. Press the **EXPORT** to display the window of [FILE] (see Figure 8-9-2). Use ↑ and ↓ keys to select the file path of the saving destination or directly enter the file path in the directory. Then, press the **ENTER** to save the exported data in the specified data file.

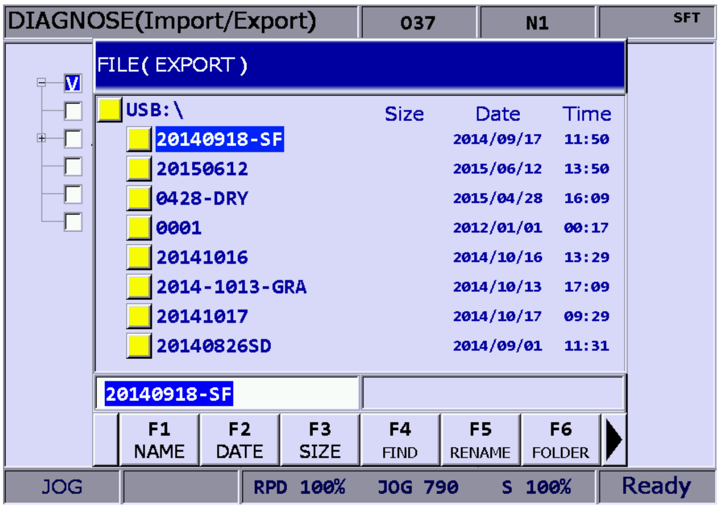


Figure 9.9.2

6. After confirmed, the exporting progress will be shown before it is complete.

7. To create a new file and save the exported data to this file (see Figure 9.9.2), please name this file and then press the **FOLDER** to save the data.

8. If the destination already contains an exported data file, a popup window will display “Update backup folder! Are you sure to execute?”. Press **Y** (yes) and then press the **ENTER** key to replace the existing data file.

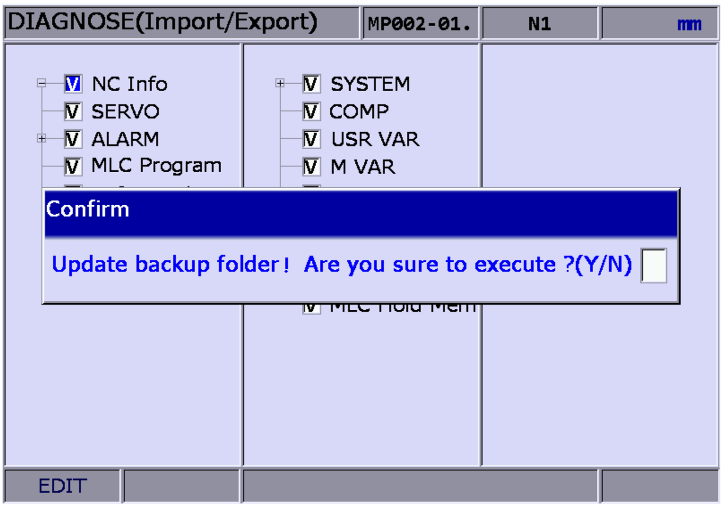


Figure 9.9.3

9. In addition, the file management function [FILE] can be operated by related function keys. (See Figure 9.9.2)

9.10 Multi language download

Language support of the group screens and function bars includes both Chinese and English. For other language support, please enhance the interface language with this multi-language downloading function. **Please contact the dealer/service provider for details.**

9.11 LOGO download [LOGO WR]

The startup screen of the system can be customized with user exclusive contents for logo presentation or other uses with this function. This function can be used only with proper permissions.

See the operation steps described below:

1. Press the **DGN** key to enter the screen of [DIAGNOSE].
2. Repeatedly press the ▶ key to enter the function bar for displaying the next page.
3. Insert a USB drive containing the correct NC300 system startup screen file.
4. Press the **LOGO WR**, a dialogue box will pop up.
5. Enter Y. It automatically accesses and loads in the start-up file from the USB.
6. Restart the system after the LOGO image file is updated.

Alarm (ALM) Group 10

The ALM group displays current alarm messages sent by the system, enabling the user to clear the program errors accordingly.

10

When an error occurs during the execution or the command format is incorrect, the system will automatically prompt alarm messages. The ALM group displays current alarm messages sent by the system for users to clear the error accordingly. Apart from displaying the current alarms, this group also keeps a record of previous alarms.

Note: Here we use **Framed Text** to indicate the keys in primary control panel. And **boldface letter** is used for indicating the function key.

10.1 Alarm

When an alarm occurs, it is required to troubleshoot the issue that has caused the alarm. Press the **RESET** key to reset the system back to the initial status. See the figure below for the alarm display. The marked area in the figure is the displaying area for alarm messages.

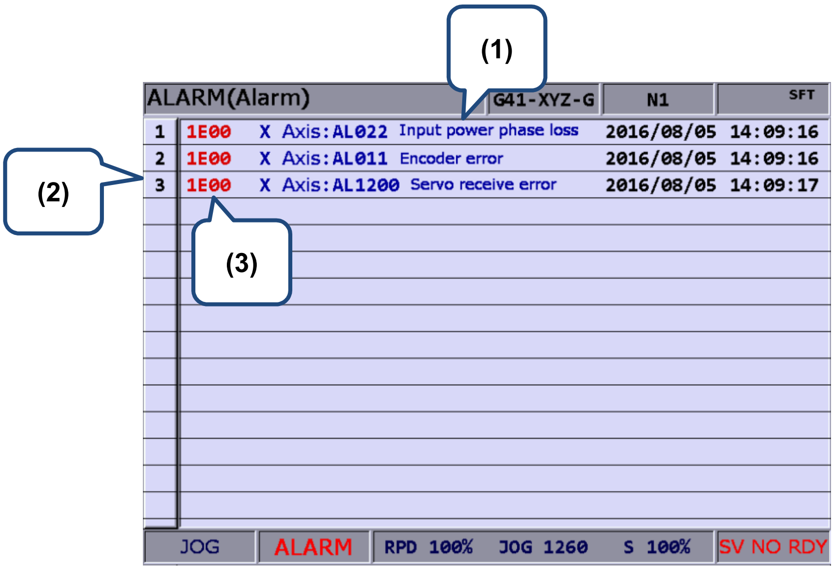


Figure 10.1.1

(1) Alarm description (3) Alarm code

(2) Sequence number of alarms

See the steps described below for the alarm message displaying and clearing:

1. Press the **ALM** key to enter the screen of [Alarm].
2. Press the **Alarm** to enter the current alarm message screen.
3. Use the **RESET** key to clear the alarm message now shown on the screen.

10.2 Alarm history

This function records alarms and the related information generated by the system. Users may review all errors during program execution by sequence of the alarm time and type for troubleshooting and analysis. Data contained in each alarm record covers the occurring time and name of alarm. This screen displays up to 512 data. Apart from displaying messages, this function enables users to clear all alarm history. See figure below.

Figure 10.2.1

See the steps described below to clear all alarm history:

1. Press the **ALM** key to enter the screen of [Alarm].
2. Press the **History** to enter the alarm history page.
3. Press the **CLR ALL** and the confirmation dialog box will pop up.
4. Press **Y** (yes) key then press the **ENTER** key and all alarm records will be removed.

(This page is intentionally left blank.)

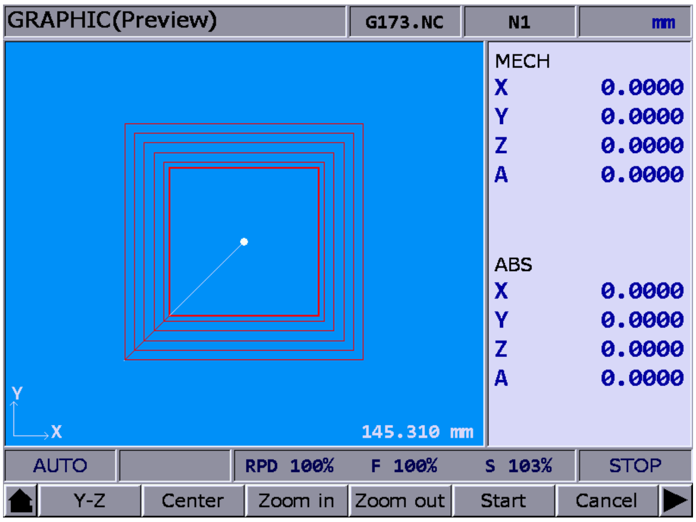
Graphic (GRA) Group 11

The GRA group displays trails during program running to help ensuring the correctness of the motion specified by G codes.

11

The GRA group displays trails during program running to help ensuring the correctness of the motion specified by G codes. It enables users to preview the machining path and perform simulation of the current machining task. Its function also includes G code format examination and machining path preview.

Note: Here we use Framed Text to indicate the keys in primary control panel. And **boldface letter** is used for indicating the function key.



11.1 Machining path (PATH)

When executing machining program, if the screen is switched to GRA group, the system will draw the motion track of the current program in the screen. When it is used for machining, this function can help to check if the path conforms to the machining task. The motion track will be drawn when the machining program is executed. And movement tracking of X-Y, Y-Z, X-Z as well as X-Y-Z plane will be provided. Users may zoom in, zoom out and moving the graph display. Use graph parameter 14003 to setup the display mode. Users may setup the positive position based on the machine type. When enabling machining path (PATH), the function of machining preview (PREVIEW) is disabled. See the operation steps below:

1. Press the GRA key to enter screen of [GRAPHIC].
2. Press the **CUTTING PATH** to the graphic display function.
3. Press the **X-Y is the initial panel** to display the movement trail for plane of X-Y; or press the same key again to for plane of Y-Z ; or press the same key again to display the plane of X-Z; or press the same key again for plane of X-Y-Z. (Turning system displays only the plane of X-Z, and thus cannot select different planes.)
4. When machining program is running, entering GRA group will start the drawing. Press the **STOP DRAW** to stop the drawing function of machining path. Press the **DRAW** to continue the drawing.
5. Press the **CENTER** to move the current motion display to the center. Press the **ZOOM IN** or the **ZOOM OUT** to magnify or minimize the display.
6. Press **UP**, **DOWN**, **LEFT** and **RIGHT** keys (on the function bar at next page) to move the graph.

11.2 Machining preview (Preview)

This function is to preview the graph of machining path. It allows users to check if the format of G code is correct and preview the motion path without actually operating the machine tools. The machining preview displays the visual angle of X-Y, Y-Z, X-Z and X-Y-Z. Users may also zoom in/out and move the graph. The related parameters are the same as described in section 11.1. When enabling this function, actual machining operation is not allowed. Function of displaying machining path and machining preview cannot be activated at the same time. When enabling the function of machining path, please cancel the preview function or press the **RESET** key. See the operation steps below:

1. Press the **GRA** key to enter screen of [GRAPHIC].
2. Press the **PREVIEW** to enter the screen for machining preview.
3. Press the **X-Y is the initial panel** to display the movement trail for plane of X-Y; or press the same key again for plane of Y-Z ; or press the same key again for plane of X-Z; or press the same key again for plane of X-Y-Z. (Turning system displays only the plane of X-Z, and thus cannot select different planes.)
4. Press the **PREVIEW** to view the machining result of G code file. Press the **CANCEL PREVIEW** to stop the preview.
5. Press the **CENTER** to move the previewing graph to the center of the displaying frame. Press the **ZOOM IN** or the **ZOOM OUT** to magnify or minimize the display.
6. Press **UP**, **DOWN**, **LEFT** and **RIGHT** keys (on the function bar at next page) to move the graph.

Notes:

1. When activating the display of machining path, function of machining preview cannot be enabled.
2. When machining preview is enabled, actual machining operation is not allowed. Function of machining path and machining preview cannot be activated at the same time; before enabling machining path, please cancel the preview or press the **RESET** key.
3. During machining preview, switching the mode will force the preview function to be canceled.
4. If the preview has been canceled, the next preview will start from the initial block when enabling again.
5. The graph of machining path and machining preview might exceed the displaying frame because of the setting of workpiece coordinates. When drawing or preview is started, if users find no path or graph displays inside the frame, please press the **CENTER** key to move the current tracking to the center of the frame.

(This page is intentionally left blank.)

Parameter (PAR) Function 12

The PAR group has included a full range of function controls in the numerical control system. This chapter illustrates parameter settings and functions.

PAR Group regulates and sets up a full range of system control and computing parameters for easy management and optimized setup. The PAR group covers the setup of parameters for operation, tool magazine, machining, spindle, mechanical, origin, compensation, and system. **After completing the parameter setting, please validate the setting according to their parameter types. There are three types: S: Power-off the servo drive; P: Power-off the system; R: Press the RESET key.**

Note: In the following parameter tables, parameters with " ▪ " symbols are for turning only. Parameters without " ▪ " symbols are for all NC series.

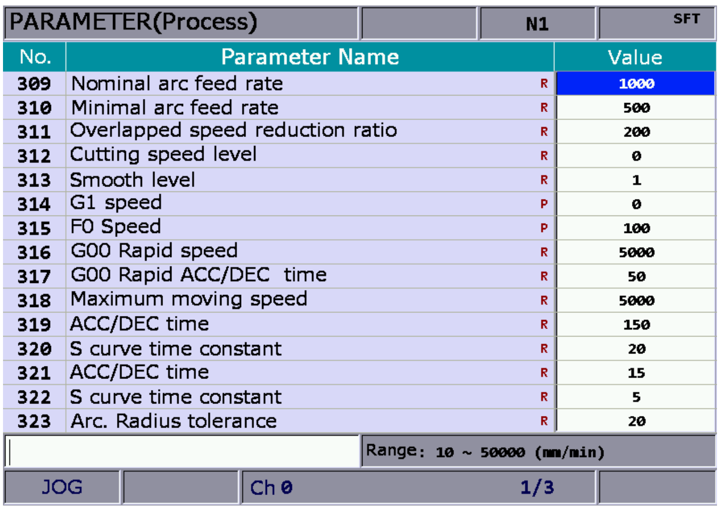
Note: Here we use **Framed Text** to indicate the keys in primary control panel. And **boldface letter** is used for indicating the function key.

12.1 Machining parameter

The machining parameter sets up parameters for the maximum cutting speed, cutting and smoothing acceleration and deceleration time. These parameters bring a huge impact on machining quality. For the best operation effect, please set up these parameters based on actual requirements.

See the operation steps below:

1. Press the **PAR** key to enter the screen of [PARAMETER].
2. Press **Process** to enter the screen of machining parameter setup.
3. Use ⬆ and ⬇ keys to move the cursor to the desired data field, and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.1.1.
4. Press the **ENTER** key to complete the setting.



PARAMETER(Process) N1 SFT

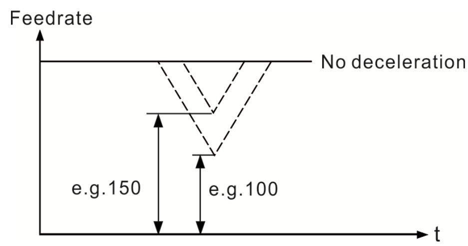
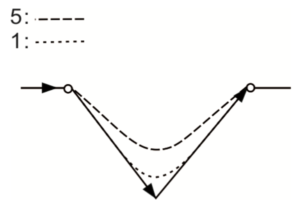
No.	Parameter Name		Value
309	Nominal arc feed rate	R	1000
310	Minimal arc feed rate	R	500
311	Overlapped speed reduction ratio	R	200
312	Cutting speed level	R	0
313	Smooth level	R	1
314	G1 speed	P	0
315	F0 Speed	P	100
316	G00 Rapid speed	R	5000
317	G00 Rapid ACC/DEC time	R	50
318	Maximum moving speed	R	5000
319	ACC/DEC time	R	150
320	S curve time constant	R	20
321	ACC/DEC time	R	15
322	S curve time constant	R	5
323	Arc. Radius tolerance	R	20

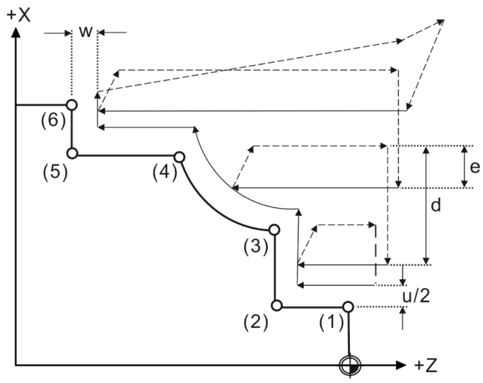
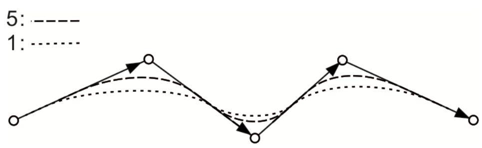
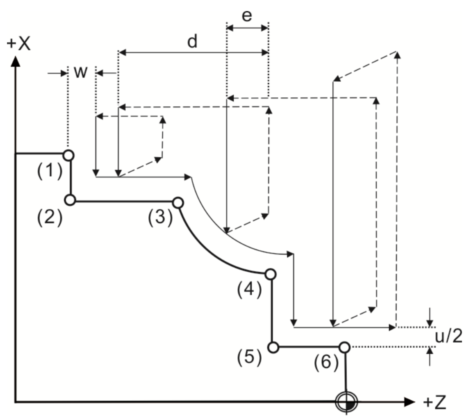
Range: 10 ~ 50000 (mm/min)

JOG Ch 0 1/3

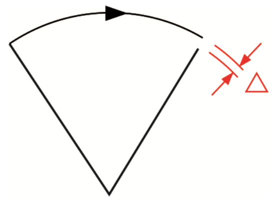
Figure 12.1.1

12.1.1 Machining parameter setting

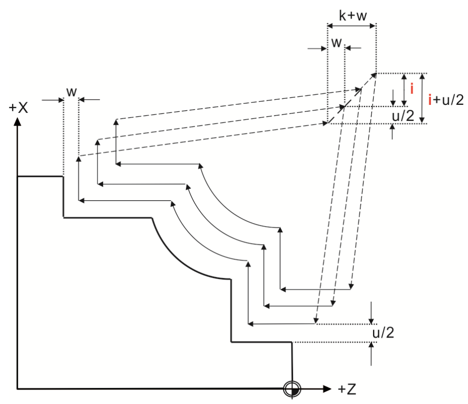
No.	Item	Description	Default	Range	Remark
309	Arc feed rate	When executing arc cutting, the arc shrinks inward because of the delayed following of the servo. If the shrinkage is fixed, you can set the arc diameter of this parameter as the max. feed rate. If the value is greater, the inward shrinkage of arc will be greater but machining precision is lower, and vice versa. Unit: mm/min	1000	10~ 50000	R
310	Minimum arc feed rate	Set up the minimum feed rate for executing arc cutting command. Unit: mm/min	500	10~ 50000	R
311	Overlapped speed reduction ratio (corner speed limit)	It sets the corner speed. When the value is greater, it is able to keep the high speed in the corners so that it's easier to return to the feedrate before deceleration. Machine vibration is more likely to occur in the corners if the setting value is too high or improper. Unit: mm/min 	100	0~50000	R
312	Level of cutting speed adjustment	It sets the level of cutting speed adjustment. 0: default Range: 1 ~ 10 In the corners, the higher the setting value is, the higher the corner speed will be; however, the machining precision will be lower. 	0	0~10	R

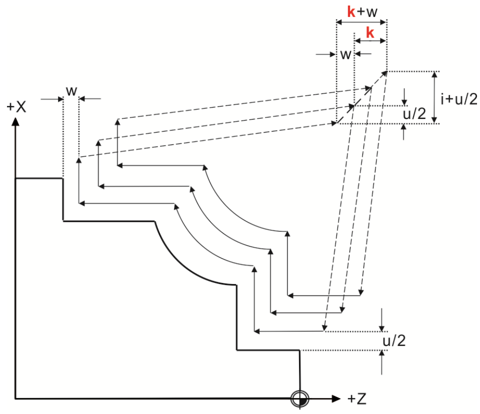
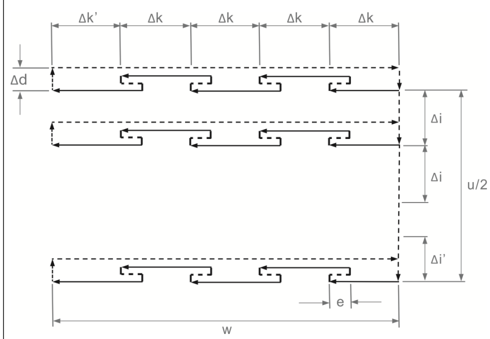
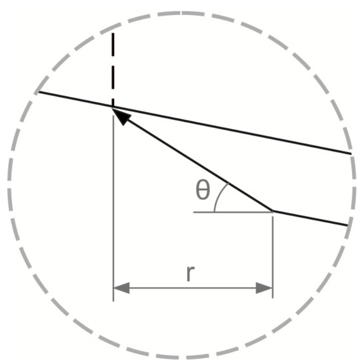
No.	Item	Description	Default	Range	Remark
▪312	Cutting depth of G71 /G72 (roughing cycle)	It sets the default cutting depth when G71 and G72 (rough turning cycle) are executed. See the figure below, **d** represents the cutting depth of G71. 	1000	0~50000	R
313	Level of smoothness adjustment	It sets the level of smoothness adjustment. 0: not smoothed Range: 1 ~ 10 When using the path smooth function, a greater value will make the path be more similar to the original one specified by the program; the precision will be higher but the speed becomes lower. 	0	0~10	R
▪313	Retraction height of G71 and G72 (roughing cycle)	It sets the retraction height when G71 and G72 (rough turning cycle) are executed. See the figure below, **e** represents the retraction height of G71. 	1000	0~50000	R

No.	Item	Description	Default	Range	Remark
314	Default cutting feed rate	It sets the default cutting speed. When this parameter value is specified, the cutting speed will refer to the parameter's setting value even when **F** value is not specified by the cutting command. Unit: mm/min, inch/min	0	0~20000	P
315	F0 speed	It sets the rapid feeding speed when the rapid factor is swiched to 0%. Unit: mm/min, inch/min	100	10~ 10000	P
316	G00 feed rate	It sets the rapid feed rate when G00's rapid factor is 100%. Unit: mm/min, inch/min	5000	1~60000	R
317	Rapid ACC/DEC time G00 acceleration/deceleration time constant	It sets the acceleration time of rapid movement; its S curve time is the same as that of parameter 319. Unit: msec	200	1~2000	R
318	Maximum moving speed	It sets the max. cutting speed. Unit: mm/min, inch/min	5000	1~60000	R
319	ACC/DEC time Cutting speed of acceleration/deceleration time constant)	It sets the acceleration time of cutting. Unit: msec (before ACC/DEC interpolation)	200	1~2000	R
320	S curve time constant (before interpolation)	It sets the S-curve time of cutting. Unit: msec (before ACC/DEC interpolation)	20	1~2000	R
321	ACC/DEC time	It sets the acceleration time of the rapid movement. The greater the value is, the more significant the profiling error will be. Unit: msec (before ACC/DEC interpolation)	50	1~500	R
322	S curve time constant (after interpolation)	It sets the acceleration/deceleration time of S curve. Unit: msec (after ACC/DEC interpolation)	10	1~100	R

No.	Item	Description	Default	Range	Remark
▪322	ACC/DEC time constant of thread turning	It sets the time required when accelerating/decelerating to the target speed during thread turning. Target speed: rpm x pitch The smaller the value is, the shorter the invalid thread's length is; but the vibration might be greater. The greater the value is, the longer the invalid thread's length will be; and vibration is minor.	10	1~100	R
323	Arc radius tolerance	 It sets the tolerance proportion of the arc radius based on the unit set in parameter 301 Unit: um	1	1~60000	R
329	Max. distance of one block for path smoothing	It sets the max. block distance of G01. When single block of G01 exceeds one block's max. distance for curve fitting, the curve fitting of this block will be canceled automatically and kee[s executing linear interpolation remains executing. Unit: 0.1 mm	20	0~10000	R
330	Max. angle for path smoothing	It sets the allowable angle for path smoothing. When the angle specified by one block exceeds the angle for curve fitting, the curve fitting for the corner will be automatically canceled and its sharpness will be kept. Unit: degree	15	0~90	R
331	Min. traveling distance of corner detection	It sets the minimum distance for corner detection. If the corner is formed of paths that are too short, the system will keep looking for the adjacent paths that are long enough and then start calculating the corner angle. Unit: 0.0001 mm	100	0~10000	R

No.	Item	Description	Default	Range	Remark
332	Tolerance of single-block path smoothing	To have a smoother machining path, this function automatically adjusts a block’s coordinates. This parameter sets the adjustable distance. The greater the value is, the smoother a single block’s path will be; the path is more unlikely to pass the coordinates set by the block. When the value is smaller, its fitting curve will be closer to the coordinates specified by the original program. Unit: 0.0001 mm	0	0~10000	R
333	Tolerance of path smoothing	It sets the tolerance of curve fitting. The greater the value is, the smoother the curve will be; however, a greater tolerance will cause low precision of the machining contour. When the tolerance is smaller, the machining curve is more likely to fit the linear interpolation path of the original program but less smoother. It is suggested that this parameter value and the error set in the CAM software to be consistent so as to make the path smoother without affecting the precision. Unit: 0.0001 mm	100	0~50000	R

No.	Item	Description	Default	Range	Remark
344	Radius of 4th axis	It sets the radius for the rotation axis. As the feeding unit of the linear axis is mm/min and rotation axis is deg/min, to apply the rotation axis for tangential speed (mm/min) applications, users can set parameter 344 to fulfill the requirement. The setting value shall be as close as the distance between cutting point and the rotation center (the radius for rotation). For example, when only the rotation axis moves, if the actual distance from center to the tool nose is 10 mm, then P344 is set to 100 (10 mm), this rotation axis' tangential speed for cutting is equal to the feed rate F. If P344 is set too small, cutting speed will be faster; If P344 is set too big, the cutting speed will become slower. Unit: 0.1 mm	0	0~2000	R (For woodworking machinery only)
▪345	Cutting depth on axis X for G73	It sets the default cutting amount in X-axis when G73 Pattern repeating cycle is executed. See the figure below. **"i"** represents the cutting volume of axis X. 	1000	0~50000	R

No.	Item	Description	Default	Range	Remark
▪346	Cutting amount on axis Z for G73	It sets the turning amount of Z-axis when G73 Pattern repeating cycle is executed. See the figure below. **K** represents the turning distance of G73. 	1000	0~50000	R
▪347	Turning time of G73	It sets the default rough turning time when G73 Pattern repeating cycle is executed.	3	1~99	R
▪348	Retraction distance of G74 / G75	It sets the default retraction distance **e** when G74 Face peck drilling cycle and G75 Axial peck drilling cycle are executed. 	1000	0~50000	R
▪349	Chamfer angle for G76 and G78	It sets the chamfer angle **θ** when G76 Thread turning cycle and G78 Single thread turning cycle are executed. 	45	1~89	R

No.	Item	Description	Default	Range	Remark
▪380	Chamfer length for G76 and G78	It sets the chamfer lengh **r** when G76 Thread turning cycle and G78 are executed. Chamfer length = setting value x 0.1 of the lead. That is to say, if **L** represents the lead, the chamfer length for thread turning will be 0L~12.7L. (Distance **r** is shown in the figure of parameter 349.)	3	0~127	R
▪381	Finish-cut time for G76	It sets the machining time of finish cut when G76 Thread turning cycle is executed.	1	1~99	R
▪382	Threading angle (infeed angle) for G76	It sets the threading angle (infeed angle) when G76 threading cycle is executed.	60	0~80	R
▪383	Min. cutting depth for G76	It sets the minimum cutting depth when G76 Thread turning cycle is executed.	1000	0~50000	R
▪439	Reserved volume of finish cut for G76	It sets the reserved amount of finish turning when G76 Thread turning cycle is executed.	200	0~50000	R

12

12.2 Operation parameter

Users can combine the execution and computing of a macro program in the G code file for composite motions. Users also can control or execute the execution of a macro program in the screen of [**PARAMTER (Operation)**].

See the operation steps below:

1. Press the **PAR** key to enter the screen of [PARAMETER].
2. Press the **Operate** to enter the operation parameter setup screen.
3. Use , , and keys to move the cursor to the desired data field, and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.2.1.
4. Press the **ENTER** key to complete the setting.

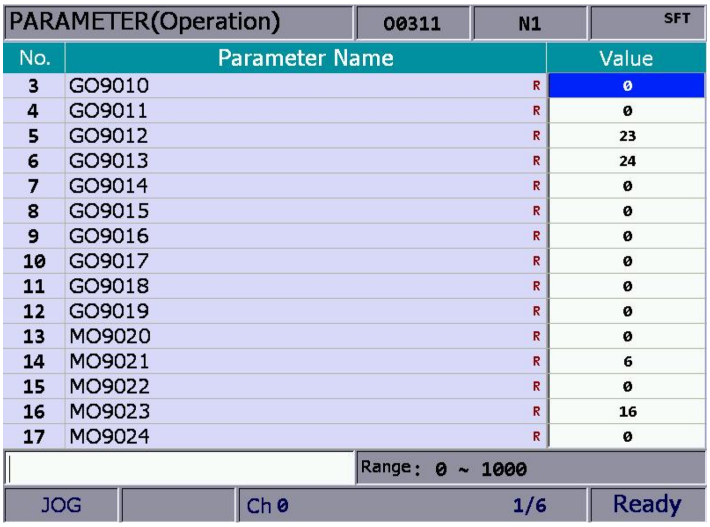


Figure 12.2.1

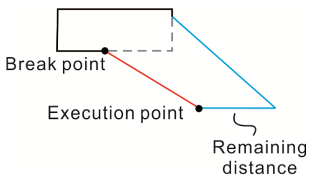
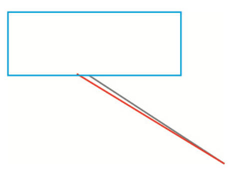
12.2.1 Operation parameter setting

No.	Name	Description	Default	Range	Remark
3~12	O9010~O9019	Set G code to call macro O9010; 0: disable the function of calling macros.	0	0~ 1000	R
		Set G code to call macro O9011			
		Set G code to call macro O9012			
		Set G code to call macro O9013			
		Set G code to call macro O9014			
		Set G code to call macro O9015			
		Set G code to call macro O9016			
		Set G code to call macro O9017			
		Set G code to call macro O9018			
		Set G code to call macro O9019			
13~22	O9020~O9029	Set M code to call macro O9020. 0: disable the function of calling macros.	0	0~ 1000	R
		Set M code to call macro O9021			
		Set M code to call macro O9022			
		Set M code to call macro O9023			
		Set M code to call macro O9024			
		Set M code to call macro O9025			
		Set M code to call macro O9026			
		Set M code to call macro O9027			
		Set M code to call macro O9028			
		Set M code to call macro O9029			
23	O9000	Set T code to call macro O9000. 0: disable the function of calling macros. 1: any T code will call a macro.	0	0~1	R
24	O9030	Call macro O9030 when executing breakpoint search. 0 (call): When breakpoint is found, carry on executing the program that followed by and perform the machining without calling the macro. 1 (not to call): After the breakpoint is found and cycle is started, call to execute O9030 first; once it is completed, return to the main program and carry on the machining program followed by the breakpoint.	0	0~1	R

<table>
<tr><th>No.</th><th>Name</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td>25</td><td>System DIO signal polarity settings</td><td>Set up DI polarity of system HSI 1/2<table><tr><th>Bit</th><th>Description</th><th>Range</th></tr><tr><td>0~1</td><td>HSI 1 (G31 interrupt) and HSI 2 settings:
0: NC contact open > H;
close > L
1: NO contact open > L;
close > H</td><td>0~3</td></tr></table></td><td>0</td><td>0~65535</td><td>P</td></tr>
<tr><td>46</td><td>System utility settings</td><td><table><tr><th>Bit</th><th>Description</th><th>Range</th></tr><tr><td>5</td><td>High speed input point (G31)
1: Enable input point of G31
0: Disable</td><td>0~1</td></tr><tr><td>10</td><td>Ignore movement command floating point
0: Do not ignore, i.e. 1 = 1 μm
1: Ignore, 1 = 1 mm</td><td>0~1</td></tr><tr><td>11</td><td>G00 operation mode
0: Rapid feeding by linear interpolation (same as G01)
1: Each axis conducts rapid feeding with the max. speed.</td><td>0~1</td></tr><tr><td>12</td><td>Macro preview function
0: Disable
1: Enable</td><td>0 ~1</td></tr></table></td><td>1100</td><td>0~ 0xFFFF</td><td>P</td></tr>
<tr><td>47</td><td>MPG gain</td><td>MPG filter gain adjusts the response of MPG. The larger the value is, the faster it reacts. But this may easily cause machine 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>MPG filter settings:
0: None<table><tr><th>Level</th><th>1</th><th>2</th><th>3</th><th>4</th><th>5</th><th>6</th></tr><tr><td>Khz</td><td>312</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>Sevo axis output settting</td><td>Ouput port setting of servo axis
0: Four axes including limit signal and homing signal
1: The positive limit, negative limit, and homing signal of the forth axis of AXIS1~4 port is set to the homing signal for axis 4, 5, and 6.</td><td>0</td><td>0 ~ 1</td><td>R</td></tr>
</table>

<table>
<tr><th>No.</th><th>Name</th><th colspan="3">Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td rowspan="3">50</td><td rowspan="3">Show macro file</td><td>Bit</td><td>Description</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>Show macro file O</td><td>0 ~ 1</td></tr>
<tr><td>1</td><td>Show G/M macro file</td><td>0 ~ 1</td></tr>
<tr><td rowspan="2">51</td><td rowspan="2">System auxiliary tool</td><td>Bit</td><td>Description</td><td>Range</td><td rowspan="2">0</td><td rowspan="2">0~1</td><td rowspan="2">P</td></tr>
<tr><td>0</td><td>Spindle operation check:
When this function is enabled, it displays alarm message when cutting command is executed but spindle is not operating.</td><td>0~1</td></tr>
<tr><td>301</td><td>Unit parameter</td><td colspan="3">Set the resolution of the coordinate value display, number of digit after the decimal mark. If it is set to 3, the unit displayed will be -99999.999 ~ 99999.999.</td><td>3</td><td>0~4</td><td>P</td></tr>
<tr><td rowspan="3">▪306</td><td rowspan="3">G code function setup</td><td>Bit</td><td>Description</td><td>Range</td><td rowspan="3">532</td><td rowspan="3">0~
0xFFFF</td><td rowspan="3">P</td></tr>
<tr><td>9~10</td><td>G code type for turning
0: Type A
1: Type B
2: Type C</td><td>0~3</td></tr>
<tr><td>11</td><td>Specify the diameter / radius of axis X
0: Diameter
1: Radius</td><td>0~1</td></tr>
<tr><td rowspan="4">307</td><td rowspan="4">Channel utility setup</td><td>Bit</td><td>Description</td><td>Range</td><td rowspan="4">0xD4</td><td rowspan="4">0~
0xFFFF</td><td rowspan="4">P</td></tr>
<tr><td>0</td><td>Moving mode of tool length compensation
0: Execute/cancel tool compensation and axis Z moves when the same block has no command on axis Z.
1: Execute/cancel tool compensation and Z axis does not move when the same block has no command on axis Z.</td><td>0~1</td></tr>
<tr><td>4~5</td><td>G31 input selection
0: input via PLC
1: HSI 1 (latch input 1)
2: HSI 2 (latch input 2)</td><td>0~1</td></tr>
<tr><td>6</td><td>Set up the returning method when breakpoint is</td><td>0~1</td></tr>
</table>

12

No.	Name	Description			Default	Range	Remark
			encountered: 0: When tool position is changed (moved) while a block is still being executed: To carry on executing the rest of the program, the cutter will go the remaining distance first and then return to the coordinate specified by the next block. See the figure below.  1: When tool position is changed (moved) while a block is still being executed: To carry on executing the rest of the program, the cutter returns to the position where its position was changed and then goes for the remaining distance. See the figure below. 				
		8~	Emergency Stop mode 0: After emergency stop, the servo swiches to Servo off state 1: After emergency stop, the servo waits and then switches to servo off state 2: After emergency stop, it remains the same servo	0~2			

<table>
<tr><th>No.</th><th>Name</th><th colspan="3">Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td rowspan="2"></td><td rowspan="2"></td><td></td><td>state (servo on).</td><td></td><td rowspan="2"></td><td rowspan="2"></td><td rowspan="2"></td></tr>
<tr><td>10</td><td>MLC variable type setting
0: word type
1: double word type
The interface input / output register; if it is set to 1, the 16 registers (16-bit) will be changed to 8 registers (32-bit).</td><td>0 ~1</td></tr>
<tr><td>324</td><td>Retraction amount of peck drilling</td><td colspan="3">Set the retraction amount of peck drilling.
UNIT: um</td><td>100</td><td>1~50000</td><td></td></tr>
<tr><td rowspan="3">326</td><td rowspan="3">Cycle parameter</td><td>Bit</td><td>Description</td><td>Range</td><td rowspan="3">0</td><td rowspan="3">0~
0xFFFF</td><td rowspan="3">R</td></tr>
<tr><td>0~1</td><td>The direction of the cyclic retraction axis
(For setting the retraction direction of G76 and G87 retraction cycle command)
0: +X 1: -X
2: +Y 3: -Y</td><td>0~3</td></tr>
<tr><td>2~3</td><td>Tapping mode
Only available on NC311/NC310)
0: General mode
1: deep pecking (feeding amount Q, retracts to R)
2: general pecking (feeding amount Q, retracting amount D)</td><td>0~2</td></tr>
<tr><td>327</td><td>EMG stop time constan</td><td colspan="3">Set up the time required for a servo motor to decelerate to stop after the emergency button is pressed.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 colspan="3">Set up the delay time of flag enabled (M2115) during emergency stop in Servo Off state. Unit: msec</td><td>35</td><td>0~2000</td><td>R</td></tr>
<tr><td>334</td><td>Rapid reduction ratio</td><td colspan="3">It sets the rapid reduction ratio when more than two G00 are applied.
When G00 is executed, the specified path including start point and stop point in a block will both be passed.</td><td>0</td><td>0~100</td><td>R</td></tr>
</table>

<table>
<tr><th>No.</th><th>Name</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td></td><td></td><td>However, if more than two G00 blocks are applied continuously, while current G00 block is almost complete and decelerates to the speed ratio (%) set by P334, the next G00 block will be executed in advance. Unit: %</td><td></td><td></td><td></td></tr>
<tr><td rowspan="10">350
~
359</td><td rowspan="10">Halt M code
1 ~10</td><td>Halt M code 1 (0: no setting)</td><td rowspan="10">0</td><td rowspan="10">0~1000</td><td rowspan="10">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>Halt M code 9</td></tr>
<tr><td>Halt M code 10</td></tr>
<tr><td>360</td><td>Synchronous control direction</td><td>Synchronous control direction:
Bit0 ~ 5: Synchronous control
X ~ C
0: same direction
1: reverse direction
<table><tr><th>Bit</th><th>Description</th><th>Range</th></tr><tr><td>0</td><td>Synchronous direction X</td><td>0~1</td></tr><tr><td>1</td><td>Synchronous direction Y</td><td>0~1</td></tr><tr><td>2</td><td>Synchronous direction Z</td><td>0~1</td></tr><tr><td>3</td><td>Synchronous direction A</td><td>0~1</td></tr><tr><td>4</td><td>Synchronous direction B</td><td>0~1</td></tr><tr><td>5</td><td>Synchronous direction C</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>Set the name of the master axis to be followed when axis X is specified as the slave. For example, to set axis Y as the axis to be followed for synchronous control, set this parameter value to 2.
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>Set the name of the master axis to be followed when axis Y is specified as the slave.
0: Disabled
1 ~ 6: X ~ C</td><td>0</td><td>0~6</td><td>P</td></tr>
<tr><td>363</td><td>Synchronous</td><td>Set the name of the master axis to be followed when</td><td>0</td><td>0~6</td><td>P</td></tr>
</table>

<table>
<tr><th>No.</th><th>Name</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td></td><td>control Z</td><td>axis Z is specified as the slave.
0: Disabled
1 ~ 6: X ~ C</td><td></td><td></td><td></td></tr>
<tr><td>364</td><td>Synchronous control A</td><td>Set the name of the master axis to be followed when axis A is specified as the slave.
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>Set the name of the master axis to be followed when axis B is specified as the slave.
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>Set the name of the master axis to be followed when axis C is specified as the slave.
0: Disabled
1 ~ 6: X ~ C</td><td>0</td><td>0~6</td><td>P</td></tr>
<tr><td>370</td><td>Transfer control direction</td><td>Transfer control direction
Bit0 ~ 5: Synchronous control X ~ C
0: same direction
1: reverse direction
<table>
<tr><th>Bit</th><th>Description</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>This parameter specifies axis X as the axis that command is tranfered to during transfer control. That is, the commanded motion will be transferred to axis X and the axis specified by the origin command remains intact. For example: To transfer the command of axis Y, set this value to 2.
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>This parameter specifies axis Y as the axis that command is tranfered to during transfer control. That is, the commanded motion will be transferred to axis Y and the axis specified by the origin command remains</td><td>0</td><td>0~6</td><td>P</td></tr>
</table>

<table>
<tr><th>No.</th><th>Name</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td></td><td></td><td>intact.
0: Disabled
1 ~ 6: X ~ C</td><td></td><td></td><td></td></tr>
<tr><td>373</td><td>Transfer control Z</td><td>This parameter specifies axis Z as the axis that command is tranfered to during transfer control. That is, the commanded motion will be transferred to axis Z and the axis specified by the origin command remains intact.
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>This parameter specifies axis A as the axis that command is tranfered to during transfer control. That is, the commanded motion will be transferred to axis A and the axis specified by the origin command remains intact.
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>This parameter specifies axis B as the axis that command is tranfered to during transfer control. That is, the commanded motion will be transferred to axis B and the axis specified by the origin command remains intact.
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>This parameter specifies axis C as the axis that command is tranfered to during transfer control. That is, the commanded motion will be transferred to axis C and the axis specified by the origin command remains intact.
0: Disabled
1 ~ 6: X ~ C</td><td>0</td><td>0~6</td><td>P</td></tr>
<tr><td>2010</td><td>HSI trigger setting</td><td><table><tr><th>Bit</th><th>Description</th><th>Range</th></tr><tr><td>0</td><td>HSI 0 trigger setting</td><td>0~1</td></tr><tr><td>1</td><td>HSI 1 trigger setting</td><td>0~1</td></tr></table>Set up rising edge (set to 0) counting and falling edge (set to 1) counting for high speed input.</td><td>0</td><td>0 ~ 65535</td><td>P</td></tr>
<tr><td>621</td><td>Rapid and maximum</td><td>Set up the maximum speed in rapid mode and manual mode.</td><td>5000</td><td>0~60000</td><td>P</td></tr>
</table>

No.	Name	Description	Default	Range	Remark
	speed	Unit: mm/min, inch/min, rpm			
622	ACC / DEC time constant	Set up the acceleration time. Unit: msec	50	0~10000	P
623	S curve time constant	Set up the S-curve time constant. Unit: msec	5	1~2000	P
635	Feed forward gain ratio	Set up the compensation ratio of the feed forwad gain.	0	0~200	
643	Allowable following error	Set up the allowable following error of the servo. If the following error exceeds the setting value during any movement, the alarm occurs. (CU : command unit)	30000	1~60000	R

12.3 Tool magazine parameter

The tool magazine parameters set up relevant functions of the tool magazine including its mechanical type, quantity, and startup. For settings of tool magazine hardware relevant parameters, please contact the dealer/service provider.

See the operation steps below:

1. Press the **PAR** key to enter the screen of [PARAMETER].
2. Press the **Maga** to enter the tool magazine parameter setup screen.
3. Use and keys to move the cursor to the desired data field, and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.3.1.
4. Press the **ENTER** key to complete the setting.

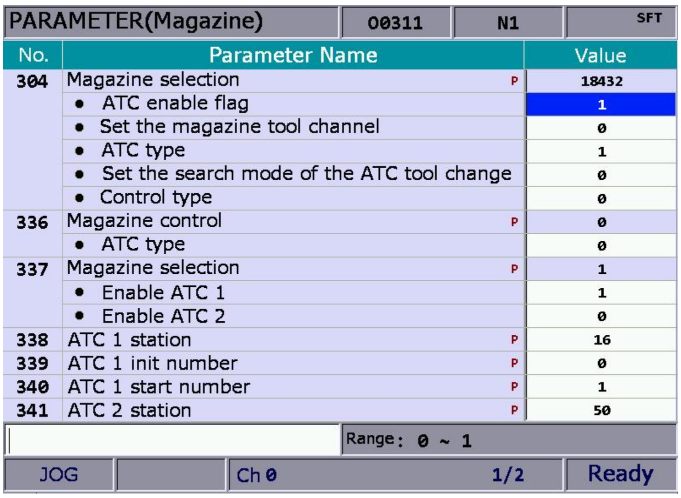


Figure 12.3.1

12.3.1 Tool magazine parameter setting

<table>
<tr><th>No.</th><th>Name</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td>304</td><td>Tool magazine parameters setup</td><td>Tool magazine channel
<table>
<tr><th>Bit</th><th>Description</th><th>Range</th></tr>
<tr><td>14</td><td>Tool magazine function
0: disable
1: enable</td><td>0~1</td></tr>
<tr><td>9</td><td>Set up the tool magazine number
0: tool magazine 0
1: tool magazine 1</td><td>0~1</td></tr>
<tr><td>12~13</td><td>Cutter search mode
0: shortest path
1: CW (reserved)
2: CCW (reserved)</td><td>0~2</td></tr>
<tr><td>15</td><td>Tool magazine control type
1: NC control</td><td>0~1</td></tr>
</table></td><td>0x4800</td><td>0~0xFFFF</td><td>P</td></tr>
<tr><td>336</td><td>Tool magazine database control</td><td><table>
<tr><th>Bit</th><th>Description</th><th>Range</th></tr>
<tr><td>11</td><td>ATC (Automatic Tool Change) type
0: Exchanger
1: Non-exchanger</td><td>0~1</td></tr>
</table></td><td>0x0800</td><td>0~65535</td><td>P</td></tr>
<tr><td>337</td><td>Tool magazine database settings</td><td><table>
<tr><th>Bit</th><th>Description</th><th>Range</th></tr>
<tr><td>0</td><td>Tool magazine database 1
0: Disable (Tool magazine data table will not be updated promptly.)
1: Enable (Tool magazine data will be promptly updated.)</td><td>0~1</td></tr>
<tr><td>1</td><td>Tool magazine database 2
0: Disable
1: Enable</td><td>0~1</td></tr>
</table></td><td>1</td><td>0~3</td><td>P</td></tr>
<tr><td>338</td><td>ATC 1 station</td><td>Set up number of stations of tool magazine 1. (This parameter is applicable for different tool magazine mechanism, users can set the placable tool number via this setting)</td><td>10</td><td>2~255</td><td>P</td></tr>
</table>

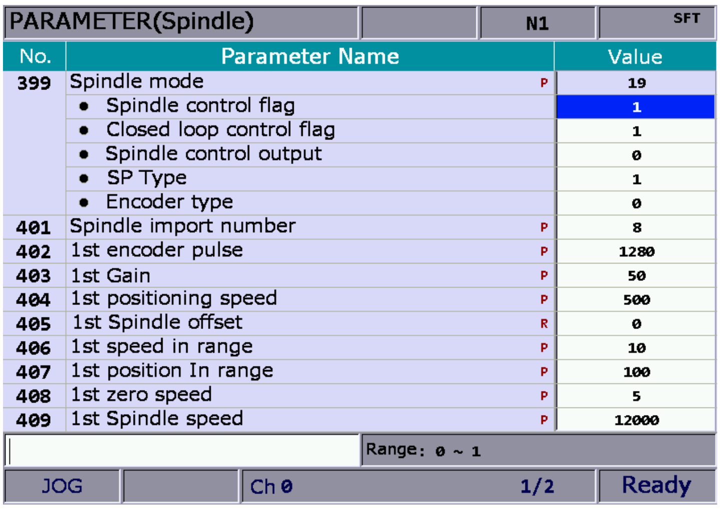
No.	Name	Description	Default	Range	Remark
339	ATC 1 init number	Set up the corresponding tool pot of tool magazine 1 after reset function is applied.	1	1~100	P
340	ATC 1 start number	Set up the start tool number of the corresponding tool pot after tool magazine system 1 is reset.	1	1~100	P
341	ATC 2 station	Set up the station number of tool magazine 2 (This parameter is applicable for different tool magazine mechanism, users can set the placable tool number via this setting)	10	2~255	P
342	ATC 2 init number	Set up the corresponding standby tool pot number after tool magazine 2 is reset.	1	1~100	P
343	ATC 2 start number	Set up the start tool number of the corresponding tool pot after tool magazine system 2 is reset.	1	1~100	P

12.4 Spindle parameter (Spindle)

The spindle parameters set up various spindle function including gains, maximum speed, and positioning errors.

See the operation steps below:

1. Press the **PAR** key to enter the screen of [PARAMETER].
2. Press the **Spindle** to enter the spindle parameter setup screen.
3. Use ↑ and ↓ keys to move the cursor to the desired data field and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.4.1.
4. Press the **ENTER** key to complete the setting.



No.	Parameter Name		Value
399	Spindle mode	P	19
	• Spindle control flag		1
	• Closed loop control flag		1
	• Spindle control output		0
	• SP Type		1
	• Encoder type		0
401	Spindle import number	P	8
402	1st encoder pulse	P	1280
403	1st Gain	P	50
404	1st positioning speed	P	500
405	1st Spindle offset	R	0
406	1st speed in range	P	10
407	1st position In range	P	100
408	1st zero speed	P	5
409	1st Spindle speed	P	12000

Figure12.4.1

12.4.1 Spindle parameter setting

<table>
<tr><th>No.</th><th>Name</th><th colspan="3">Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td>398</td><td>Default spindle speed</td><td colspan="3">Setting of default spindle speed.</td><td>0</td><td>0~60000</td><td>P</td></tr>
<tr><td rowspan="6">399</td><td rowspan="6">Spindle mode</td><td>Bit</td><td>Description</td><td>Range</td><td rowspan="6">0</td><td rowspan="6">0~
0xFFFF</td><td rowspan="6">P</td></tr>
<tr><td>0</td><td>Spindle function
0: Spindle Off
1: Spindle On</td><td>0~1</td></tr>
<tr><td>1</td><td>Close loop control flag
0: Close loop control Off
1: Close loop control On (requires feedback encoder)</td><td>0~1</td></tr>
<tr><td>2~3</td><td>Spindle output mode
0: DMCNET (Servo spindle)
1: DDA or DAC port
2: EDAC (analog output)</td><td>0~2</td></tr>
<tr><td>4</td><td>Speed mode
1: DMCNET (spindle communication mode)</td><td>0~1</td></tr>
<tr><td>5</td><td>Selection for spindle encoder type
0: high resolution (x1000)
1: regular resolution (x4)</td><td>0~1</td></tr>
<tr><td>401</td><td>Spindle input port number</td><td colspan="3">Set up feedback channel of spindle port for the encoder.</td><td>8</td><td>0~8</td><td>P</td></tr>
<tr><td>402</td><td>Pulse number of spindle encoder</td><td colspan="3">Set up the pulse number of encoder.
Unit: pulse/rev</td><td>1280</td><td>2~10000</td><td>P</td></tr>
<tr><td>403</td><td>Spindle gains</td><td colspan="3">Set up speed regulator gains (the smaller the value is the faster the response will be).</td><td>50</td><td>1~5000</td><td>P</td></tr>
</table>

No.	Name	Description	Default	Range	Remark
404	Spindle positioning speed	Set up the max. speed of spindle positioning. When the current speed is higher than the setting value of P404, the spindle will decelerate to the speed specified by P404 and commence positioning. Please refer to description of P420 if the spindle speed is slower than setting value of P404. Unit: rpm	100	1~20000	P
405	Spindle positioning offset	Set up positioning offset of the servo spindle. Unit: 0.01 degree	0	0~36000	R
406	Spindle target speed error	Set up allowable tolerance between spindle's nominal and actual speeds.	10	0~100	P
407	Spindle positioning error	Set up spindle positioning tolerance. Unit: 0.01 degree	100	0~36000	P
408	Range of spindle zero speed	The zero speed signal of spindle will be on when its speed is within the setting range **(NC > MLC M2257).**	5	0~1000	P
409	Spindle maximum speed	Set up maximum speed of the spindle. Unit: rpm	20000	0~50000	P
▪410	Spindle minimum speed	Set up minimum speed of the spindle. Unit: rpm	10	0~10000	P
411	Spindle acceleration time	Set up the acceleration time of the spindle. The greater the value is, the longer the acceleration/deceleration time will be. Unit: msec	20	1~20000	R
412	Spindle S curve time constant	Set up S curve time of the spindle. Unit: msec	10	1~2000	R
416	Tapping acceleration/deceleration time constant	Set up spindle acceleration / deceleration time for tapping Unit: msec	2000	1~20000	R
417	Tapping S curve time constant	Set up spindle S curve time for tapping Unit: msec	100	1~2000	R
418	Forward gain ratio of spindle	Set up the compensation ratio.	0	0~200	

No.	Name	Description	Default	Range	Remark
420	Spindle positioning low speed	The system will refer to this speed setting to do spindle positioning when the spindle is at 0 speed or at the speed slower than the setting value of P404. Unit: rpm	100	1~20000	P
421	Spindle retrieve ratio	Reserved	10	10~ 50000	R
422	Gear ratio numerator 1	Set the numerator of the spindle gear ratio (speed in first gear)	1	0~60000	P
423	Gear ratio denominator 1	Set the denominator of the spindle gear ratio (speed in first gear)	1	0~60000	P
424	Gear ratio numerator 2	Set the numerator of the spindle gear ratio (speed in second gear)	1	0~60000	P
425	Gear ratio denominator 2	Set the denominator of the spindle gear ratio (speed in second gear)	1	0~60000	P
426	Gear ratio numerator 3	Set the numerator of the spindle gear ratio (speed in third gear)	1	0~60000	P
427	Gear ratio denominator 3	Set the denominator of the spindle gear ratio (speed in third gear)	1	0~60000	P
428	Gear ratio numerator 4	Set the numerator of the spindle gear ratio (speed in fourth gear)	1	0~60000	P
429	Gear ratio denominator 4	Set the denominator of the spindle gear ratio (speed in fourth gear)	1	0~60000	P

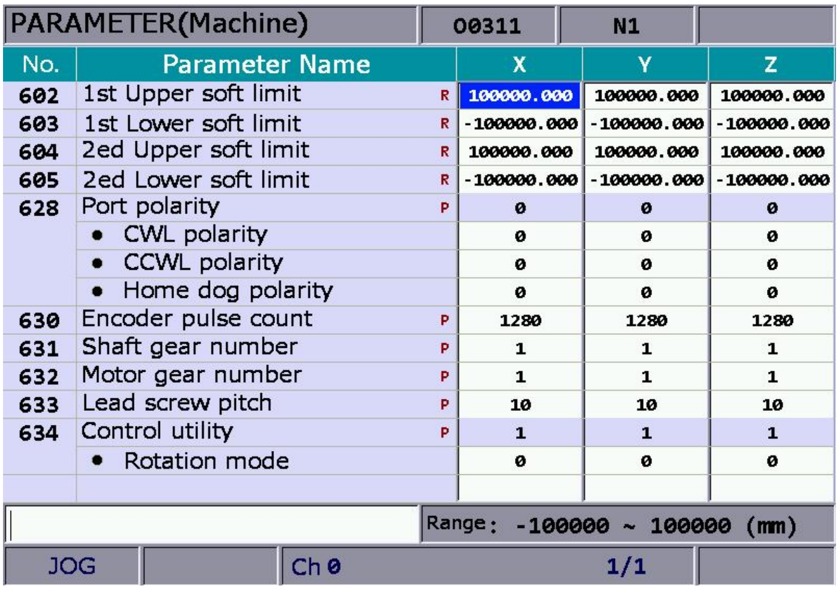
12.5 Mechanical parameter (Machine)

12

Users can set up the mechanical equipment relevant parameters of software/hardware limit, screw guide pitch and number of pulses of encoder.

See the operation steps below:

1. Press the **PAR** key to enter the screen of [PARAMETER].
2. Press the **Machine** to enter the mechanical parameter setup screen.
3. Use ↑ and ↓ keys to move the cursor to the desired data field and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.5.1.
4. Press the **ENTER** key to complete the setting.



PARAMETER(Machine)		O0311	N1	
No.	Parameter Name	X	Y	Z
602	1st Upper soft limit R	100000.000	100000.000	100000.000
603	1st Lower soft limit R	-100000.000	-100000.000	-100000.000
604	2ed Upper soft limit R	100000.000	100000.000	100000.000
605	2ed Lower soft limit R	-100000.000	-100000.000	-100000.000
628	Port polarity P	0	0	0
	• CWL polarity	0	0	0
	• CCWL polarity	0	0	0
	• Home dog polarity	0	0	0
630	Encoder pulse count P	1280	1280	1280
631	Shaft gear number P	1	1	1
632	Motor gear number P	1	1	1
633	Lead screw pitch P	10	10	10
634	Control utility P	1	1	1
	• Rotation mode	0	0	0

Range: -100000 ~ 100000 (mm)

JOG | Ch 0 | 1/1

Figure 12.5.1

12.5.1 Mechanical parameter setting

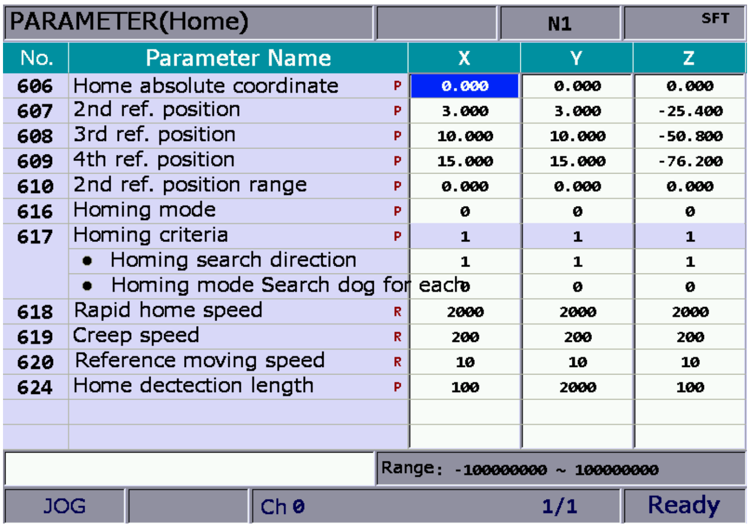
<table>
<tr><th>No.</th><th>Item</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td>602</td><td>First positive soft limit</td><td>Set up the mechanical coordinates of the first positive software limit. When it is set to 0, it means the function is disabled. Unit: mm
1. Positive software limit alarm occurs when the set limit is reached.
2. It can be specified by special M.</td><td>10^5</td><td>-10^5 ~ +10^5</td><td>R</td></tr>
<tr><td>603</td><td>First negative soft limit</td><td>Set up the mechanical coordinates of the first negative software limit. When it is set to 0, it means the function is disabled. Unit: mm
1. Negative software limit alarm occurs when the set limit is reached.
2. It can be specified by special M.</td><td>-10^5</td><td>-10^5 ~ +10^5</td><td>R</td></tr>
<tr><td>604</td><td>Second positive soft limit</td><td>Set up the mechanical coordinates of the second positive software limit. When it is set to 0, it means the function is disabled.
Positive software limit alarm occurs when the set limit is reached. Unit: mm</td><td>10^5</td><td>-10^5 ~ +10^5</td><td>R</td></tr>
<tr><td>605</td><td>Second negative soft limit</td><td>Set up the mechanical coordinates of the second negative software limit. When it is set to 0, it means the function is disabled.
Negative software limit alarm occurs when the set limit is reached. Unit: mm</td><td>-10^5</td><td>-10^5 ~ +10^5</td><td>R</td></tr>
<tr><td>628</td><td>Polarity of hard limit setting</td><td>Set up the input polarity of positive/negative hardware limit and the origin.
When it is set to 1, the contact is NO;
When it is set to 0, the contact is NC.<table><tr><th>Bit</th><th>Description</th><th>Range</th></tr><tr><td>0</td><td>Polarity setting of positive limit</td><td>0~1</td></tr><tr><td>1</td><td>Polarity setting of negative limit</td><td>0~1</td></tr><tr><td>2</td><td>Polarity setting of the origin</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 pulse number</td><td>Set up the pulse number of each motor revolution.</td><td>1280</td><td>10~ 50000</td><td>P</td></tr>
</table>

<table>
<tr><th>No.</th><th>Item</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td>631</td><td>Number of teeth of spindle</td><td>Set up the teeth number for the transmission axis. For instance, if P631 is set to 10 and P632 is set to 1 (motor/spindle = 10 turns / 1 turn; reduction ratio is 10:1), it is deceleration.
If acceleration or deceleration setting is not required, set P631 and P632 to 1.</td><td>1</td><td>1~65535</td><td>P</td></tr>
<tr><td>632</td><td>Number of teeth of motor</td><td>Set up the teeth number for the motor. See the description of P631.</td><td>1</td><td>1~65535</td><td>P</td></tr>
<tr><td>633</td><td>Lead screw pitch</td><td>Set up the lead screw pitch of the driving shaft. This setting is only valid when using linear axis such as X Y Z or set A as the linear axis.
Unit: mm</td><td>10</td><td>2~100</td><td>P</td></tr>
<tr><td>634</td><td>Axis control variables</td><td><table><tr><th>Bit</th><th>Description</th><th>Range</th></tr><tr><td>1~ 3</td><td>Feeding mode of the rotation axis (This is only applicable to rotation axis A, B, and C; not applicable to axis X, Y, and Z)
0: Specify the axis as the rotation axis and do the feeding without following the shortest path).
1: Specify the axis as the rotation axis and follow the shortest path.
2: Specify the axis as the rotation axis and display the coordinates in linear manner.
5: Specify the axis as the linear axis</td><td>0~5</td></tr></table></td><td>5</td><td>0~65535</td><td>P</td></tr>
</table>

12.6 Origin parameter (Home)

The origin parameter sets up coordinates from mechanical origin to the fourth reference point and origin searching mode.
See the operation steps below:

1. Press the **PAR** key to enter the screen of [PARAMETER].
2. Press the **Home** to enter the origin parameter setup screen.
3. Use ↑ and ↓ keys to move the cursor to the desired data field and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.6.1.
4. Press the **ENTER** key to complete the setting.


PARAMETER(Home) N1 SFT

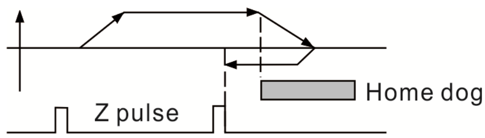
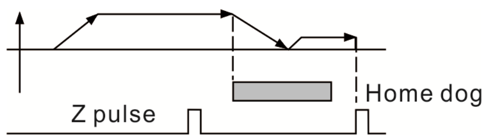
No.	Parameter Name		X	Y	Z
606	Home absolute coordinate	P	0.000	0.000	0.000
607	2nd ref. position	P	3.000	3.000	-25.400
608	3rd ref. position	P	10.000	10.000	-50.800
609	4th ref. position	P	15.000	15.000	-76.200
610	2nd ref. position range	P	0.000	0.000	0.000
616	Homing mode	P	0	0	0
617	Homing criteria	P	1	1	1
	● Homing search direction		1	1	1
	● Homing mode Search dog for each		0	0	0
618	Rapid home speed	R	2000	2000	2000
619	Creep speed	R	200	200	200
620	Reference moving speed	R	10	10	10
624	Home dectection length	P	100	2000	100

Range: -100000000 ~ 100000000

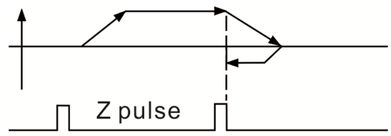
JOG Ch 0 1/1 Ready

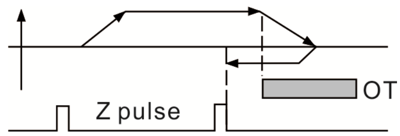
Figure 12.6.1

12.6.1 Homing origin parameter setting

No.	Item	Description	Default	Range	Remark
606	Mechanical origin coordinates (offset)	Figure out the distance form the machine's home sensor to the Z pulse. And set this distance (coordinates) as the mechanical origin coordinates. Unit: CU	0	-10^5 ~10^5	P
607	Second reference coordinates	Set up coordinates of the second reference point (G30 position setup) Unit: CU	0	-10^5 ~10^5	P
608	Third reference coordinates	Set up coordinates of the third reference point Unit: CU	0	-10^5 ~10^5	P
609	Fourth reference coordinates	Set up coordinates of the fourth reference point Unit: CU	0	-10^5 ~10^5	P
610	Second reference position error settings	Set up the position tolerance of the second reference point. For example, when the value is set to 0.2, it means the tolerance is ±0.2 mm; the second reference point can be found in this range.	0	-10^8 ~10^8	P
616	Homing mode	0: Homing disabled 1: Method 1 When touching the block (Home dog), move in reverse direction to look for Z pulse and regard it as the origin.  2: Method 2 When touching the block, move in forward direction to look for Z pulse and regard it as the origin.  3: Method 3 Move to Z pulse at low speed.	1	0~17	P

12

<table>
<tr><th>No.</th><th>Item</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td></td><td></td><td>

4: Method 4 (OT mode)
Regard the hardware limit as the home sensor in homing mode. In other modes, regard the hardware limit as Home dog.


5: Method for absolute type motor</td><td></td><td></td><td></td></tr>
<tr><td>617</td><td>Searching the origin</td><td><table>
<tr><th>Bit</th><th>Description</th><th>Range</th></tr>
<tr><td>0</td><td>Direction when looking for the origin
0: Reverse direction
1: Forward direction</td><td>0~1</td></tr>
<tr><td>1</td><td>Always look for the Home dog when homing
0: Disable
1: Enable</td><td>0~1</td></tr>
</table></td><td>1</td><td>0~7h</td><td>P</td></tr>
<tr><td>618</td><td>First stage speed of homing</td><td>Set up the first stage speed for searching the home sensor.
Unit: mm/min</td><td>2000</td><td>0~10000</td><td>R</td></tr>
<tr><td>619</td><td>Second stage speed of homing</td><td>Set up the speed for searching the Z pulse.
Unit: mm/min</td><td>200</td><td>0~2000</td><td>R</td></tr>
<tr><td>620</td><td>Reference point movement speed</td><td>The first homing procedure after start-up requires referring to the speed sepcified by P618 and P619. After the first homing procedure, the homing speed can be specified by this parameter. Unit: mm/min</td><td>10</td><td>0~20000</td><td>R</td></tr>
<tr><td>624</td><td>Home detection length</td><td>When searching the Home dog, this parameter sets the allowable distance from the Home dog. The alarm occurs when it exceeds this setting range.
Unit: mm</td><td>20</td><td>1~2000</td><td>P</td></tr>
</table>

12.7 Network Setup (Ethernet)

12

This function enables users to remotely connect to a PC through Ethernet communication. Working together with CNC Network software and the network setting of the NC numerical control system, users can use one PC to control multiple NC controllers for online file management, file sharing, file management and transmission-along-with-machining (DNC) through remote network communication.
See the operation steps below:

1. Press the PAR key to enter the screen of [PARAMETER].
2. Press the ▶ key to switch to the screen with function bar.
3. Press the **Network Set** to enter the network setup page.
4. Use ⇧ and ⇩ keys to move the cursor to the desired data field and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.7.1.
5. Press the ENTER key to complete the setting.

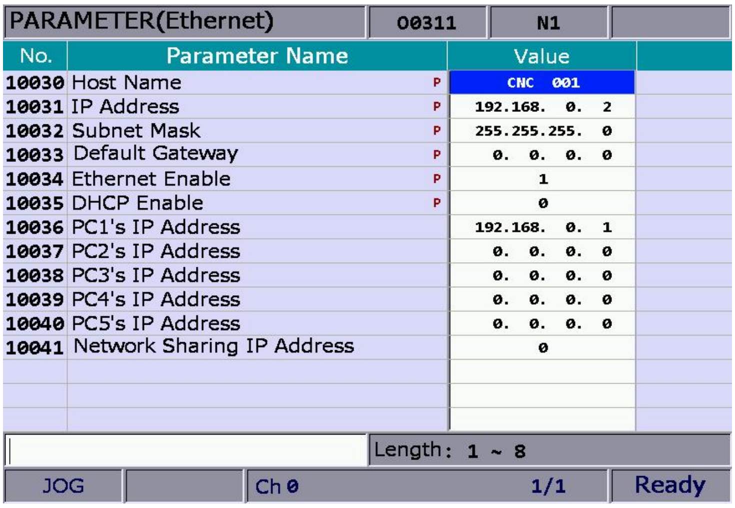


Figure 12.7.1

12

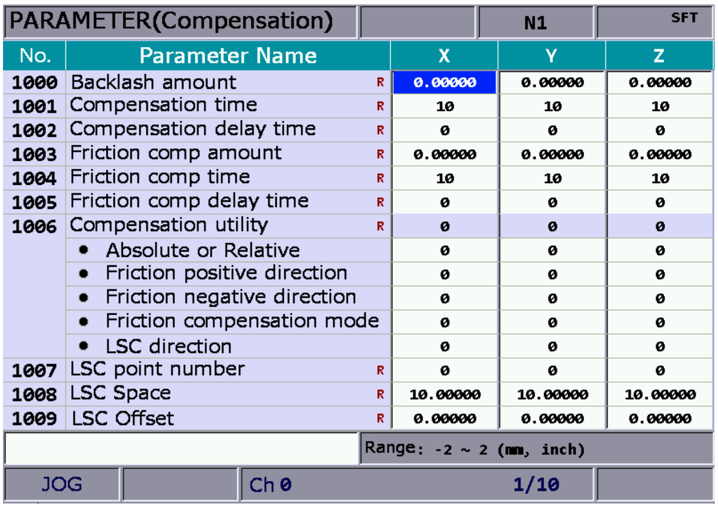
12.7.1 Network parameters setting

No.	Item	Description	Default	Range	No.
10030	Host name	Set up the host name.	CNC000	1~8	
10031	IP Address	Set up system IP address.	0.0.0.0	0~255	P
10032	Subnet mask	Set up system subnet mask.	0.0.0.0	0~255	P
10033	Default gateway	Set up system default gateway.	0.0.0.0	0~255	P
10034	Enable Ethernet function	Set up system network function 0: Disable 1: Enable	0	0~1	P
10035	Enable DHCP function	Set up DHCP function 0: Disable 1: Enable	0	0~1	P
10036 ~ 10040	Remote PC IP Address 1 ~ 5	IP address 1	0	255	
		IP address 2	0	255	
		IP address 3	0	255	
		IP address 4	0	255	
		IP address 5	0	255	
10041	IP address 1 for remote directory sharing	Edit the computer IP address specified by Network in the directory. 0: Disable the specified IP address	0	0~5	P

12.8 Compensation parameter

The compensation parameter sets up relevant compensation data to compensate errors caused by mechanical factors during actual machine operation. That is to say, the compensation is given by the control system with considering the machine features.
See the operation steps below:

1. Press the PAR key to enter the screen of [PARAMETER].
2. Press the ▶ key to switch to the screen with function bar.
3. Press the **Comp** to enter the compensation parameter setup screen.
4. Use ⬆ and ⬇ keys to move the cursor to the desired data field and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.8.1.

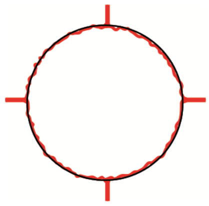
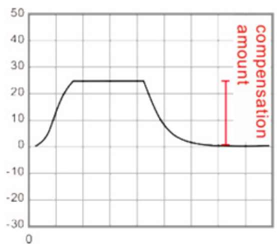
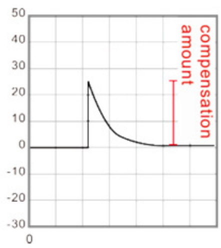
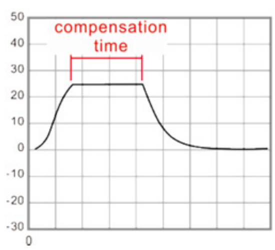
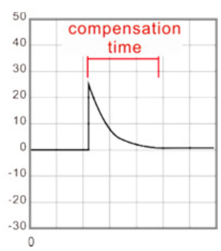


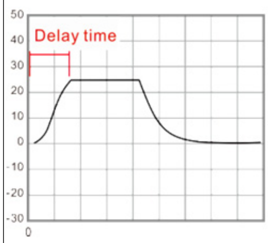
PARAMETER(Compensation)			N1	SFT
No.	Parameter Name	X	Y	Z
1000	Backlash amount R	0.00000	0.00000	0.00000
1001	Compensation time R	10	10	10
1002	Compensation delay time R	0	0	0
1003	Friction comp amount R	0.00000	0.00000	0.00000
1004	Friction comp time R	10	10	10
1005	Friction comp delay time R	0	0	0
1006	Compensation utility R	0	0	0
	• Absolute or Relative	0	0	0
	• Friction positive direction	0	0	0
	• Friction negative direction	0	0	0
	• Friction compensation mode	0	0	0
	• LSC direction	0	0	0
1007	LSC point number R	0	0	0
1008	LSC Space R	10.00000	10.00000	10.00000
1009	LSC Offset R	0.00000	0.00000	0.00000
	Range: -2 ~ 2 (mm, inch)			
JOG		Ch 0	1/10	

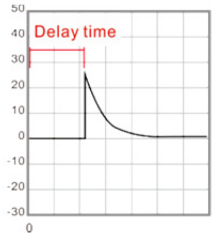
Figure 12.8.1

5. Press the ENTER key to complete the setting.
6. Press the **um** to complete the setting of absolute length compensation in unit of um; or press the **um+** to complete the setting of incremental length compensation in unit of um.
7. Compensation data generated by calibration instruments can be converted by the CNC SOFT program. Press the **import** to import compensation data in absolute type; or press the **import +** to import compensation data in incremental type.
8. After entering all compensation value, press the **OK** to confirm and update the compensation parameters.

12.8.1 Compensation parameter setting

No.	Item	Description	Default	Range	Remark
1000	Rear gap compensation value	Set up the backlash amount that is fixed in most screws of the mechanical system. Set positive value for forward backlash and negative for backward one. If the parameter value is set to 0, compensation is Off. Unit: (mm, inch)  Arc contouring example	0	-2~2	R
1001	Backlash compensation time	Set the time constant for compensation ratio for the movement direction of backlash compensation. It is effective only when backlash compensation value is given. Unit: 0.1 msec	0	0~ 10000	R
1002	Backlash compensation delay time	Set up time delay for startup compensation. Unit: 0.1 msec	0	0~ 10000	R
1003	Friction compensation amount	Friction compensation amount (mm)  Friction compensation mode = 0  Friction compensation mode = 1	0	0~1	R
1004	Friction compensation time	Friction compensation time (0.1 msec)  Friction compensation mode = 0  Friction compensation mode = 1	0	0~ 10000	R

<table>
<tr><th>No.</th><th>Item</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td>1005</td><td>Friction compensation delay time</td><td>Friction compensation delay time (0.1 msec)


Friction compensation mode = 0


Friction compensation mode = 1</td><td>0</td><td>0~10000</td><td>R</td></tr>
<tr><td>1006</td><td>Pitch compensation application setup</td><td><table>
<tr><th>Bit</th><th>Description</th><th>Range</th></tr>
<tr><td>0</td><td>Absolute or incremental input
0: Absolute input
(the actual value of the measuring point)
1: Incremental input
(the incremental value from the current to the previous measuring point)</td><td>0~1</td></tr>
<tr><td>2</td><td>Make friction compensation when machine tool moves in forward direction.</td><td>0~1</td></tr>
<tr><td>3</td><td>Make friction compensation when machine tool moves in reverse direction.</td><td>0~1</td></tr>
<tr><td>4</td><td>Mode selection for friction compensation</td><td>0~1</td></tr>
<tr><td>15</td><td>LSC direction;
Direction of the mechanical coordinates
(Measuring direction from starting point)
0: measuring toward forward direction
1: measuring toward backward direction</td><td>0~1</td></tr>
</table></td><td>0</td><td>0~
0xFFFF</td><td>R</td></tr>
<tr><td>1007</td><td>Measuring points</td><td>Set up the measuring point number for pitch compensation of the lead screw. Maximum is 128 points. If the value is set to 0, the compensation is turned off.</td><td>0</td><td>0~128</td><td>R</td></tr>
</table>

12

No.	Item	Description	Default	Range	Remark
1008	Measuring intervals	Set up the interval between each measuring points. Unit: mm	0	0~300	R
1009	Measuring offsets	Set up the offset amount from machine origin to the measuring point. For example, setting the value to 0 means the offset is 0; setting the value to 10 mm means the offset is 10 mm. Please note that the direction has to be consistent with the direction specified by Bit 15 of parameter 1006.	0	-1000~ 1000	R
1010 ~ 1137	Data 1 ~ data 128	Set up the lead screw pitch compensation of the 1st ~ 128th point. The 1st point has to be set as the same as the origin. Unit: linear axis (mm); rotation axis (deg)	0	-20~20	R

12.9 System parameter (System)

The system parameters enable users to customize operation environment including system date, system time, background color of screen, function bar color, and tab color. See the operation steps below:

1. Press the **PAR** key to enter the screen of [PARAMETER].
2. Press the ▶ key to switch to the screen with function bar.
3. Press the **System** to enter the system parameter setup screen.
4. Use ⬆ and ⬇ keys to move the cursor to the desired data field and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.9.1.

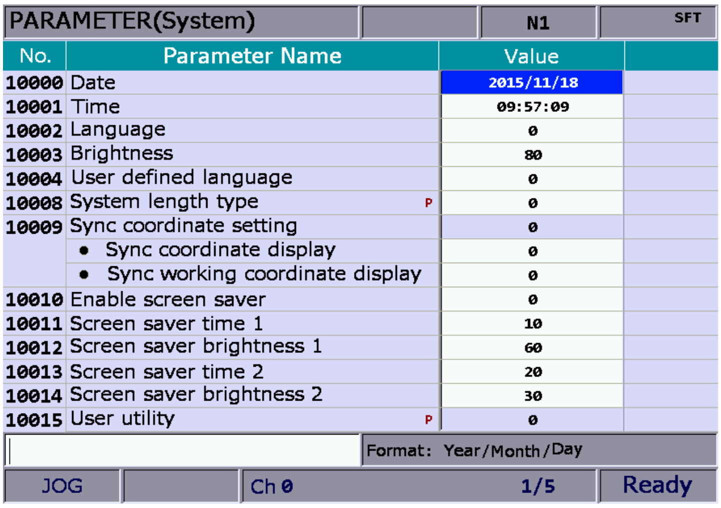


Figure 12.9.1

5. Press the **ENTER** key to complete the setting.
6. As for the setting of color items, press the **Color** and the color selection dialog box will pop up.
7. To reset the system environment back to its factory defaults status, press the **Default** and a confirmation dialog box will pop up.
8. Press “Y” (Yes) and the **ENTER** key to reset the system back to its factory defaults status.

12.9.1 System parameter setting

<table>
<tr><th>No.</th><th>Name</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td>10000</td><td>System date</td><td>Set up system date.
Format: yyyy/mm/dd</td><td></td><td></td><td></td></tr>
<tr><td>10001</td><td>System time</td><td>Set up system time.
Format: hh:mm:ss</td><td></td><td></td><td></td></tr>
<tr><td>10002</td><td>System language</td><td>System language setting
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>Set up screen brightness.</td><td>50</td><td>1~99</td><td></td></tr>
<tr><td>10004</td><td>User defined language</td><td>User defined system language</td><td>0</td><td>0~1</td><td></td></tr>
<tr><td>10008</td><td>System length type</td><td>Set up the legth to metrics/ imperial unit.
0: Metrics
1: Impreial</td><td>0</td><td>0~1</td><td>P</td></tr>
<tr><td>10009</td><td>Synchronous coordinate display</td><td>Show coordinates when synchronous function is applied
<table>
<tr><th>Bit</th><th>Desciption</th><th>Range</th></tr>
<tr><td>0</td><td>Display synchronous coordinates
0: Disable
1: Enable</td><td>0~1</td></tr>
<tr><td>2</td><td>Display workpiece coordinates during synchronous control
0: Disable
1: Enable</td><td>0~1</td></tr>
</table></td><td>0</td><td>0~65535</td><td></td></tr>
<tr><td>10010</td><td>Screen saver ON</td><td>Screen saver function
0: Off
1: On</td><td>0</td><td>0~1</td><td></td></tr>
<tr><td>10011</td><td>Time of first stage screen saver</td><td>When screen saver is On, set up the time of first stage screen saver.</td><td>10</td><td>1~60</td><td></td></tr>
<tr><td>10012</td><td>Brightness of first stage screen saver</td><td>When screen saver is On, set up the brightness of first stage screen saver.</td><td>30</td><td>0~99</td><td></td></tr>
<tr><td>10013</td><td>Time of second stage screen saver</td><td>When screen saver is On, set up the time of second stage screen saver.</td><td>30</td><td>1~60</td><td></td></tr>
<tr><td>10014</td><td>Brightness of second stage screen saver</td><td>When screen saver is On, set up the brightness of second stage screen saver.</td><td>10</td><td>0~99</td><td></td></tr>
</table>

12

<table>
<tr><th>No.</th><th>Name</th><th colspan="3">Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td rowspan="4">10015</td><td rowspan="4">User related setting</td><td colspan="3">User related setting</td><td rowspan="4">0</td><td rowspan="4">0~65535</td><td rowspan="4">P</td></tr>
<tr><td>Bit</td><td>Description</td><td>Range</td></tr>
<tr><td>1</td><td>Apply the way the page of [User 1] is opened
0: System built-in (password)
1: External I/O
M2934 = 1 (Lock)
M2934 = 0 (Unlock)</td><td>0~1</td></tr>
<tr><td>2</td><td>Automatically open the file used last time. When this function is enabled, the system will open the file that is used last time as soon as the USB flash drive or CF card is plugged in.
0: Disable
1: Enable</td><td>0~1</td></tr>
<tr><td rowspan="5">10016</td><td rowspan="5">System related setting</td><td colspan="3">System related setting:</td><td rowspan="5">4</td><td rowspan="5">0~65535</td><td rowspan="5">P</td></tr>
<tr><td>Bit</td><td>Description</td><td>Range</td></tr>
<tr><td>0</td><td>Reset the system after EMG stop is released. (Set up whether to generate a Reset signal after the EMG stop is released.)
0: Disable
1: Enable</td><td>0~1</td></tr>
<tr><td>1</td><td>Display the default software panel when startup
0: Off
1: On</td><td>0~1</td></tr>
<tr><td>2</td><td>Pop up the alarm screen when an alarm occurs
0: Off</td><td>0~1</td></tr>
</table>

No.	Name	Description			Default	Range	Remark
			1: On				
		3	Auto-back up the system parameter (When this function is enabled, the system will automatically back up the parameter data to CF card. As long as the parameter data is changed, the data in CF card will be changed.) 0: Disable 1: Eanble	0~1			
		4	Hide the axis coorniates In [Channel setting], if you have selected to hide the axis, the corresponding axis coordinates will also be hidden.) 0: Hide axis coordinates 1: Show axis coordinates	0~1			
		5	O Macro file protection When this function is enabled, file can only be copied within internal of O Macro file; copying file to the external device is not allowed) 0: Disable 1: Enable	0~1			
		8	POS group display 0: Enable 1: Disable	0~1			
		9	PRG group display 0: Enable 1: Disable	0~1			

No.	Name	Description			Default	Range	Remark
		10	OFS group display 0: Enable 1: Disable	0~1			
		11	DGN group display 0: Enable 1: Disable	0~1			
		12	ALM group display 0: Enable 1: Disable	0~1			
		13	GRA group display 0: Enable 1: Disable	0~1			
		14	PAR group display 0: Enable 1: Disable	0~1			
		15	SOFT group display 0: Enable 1: Disable	0~1			
10017	Open G code editor	Bit	Description	Range	1	0~65535	
		0	G code editing (You can set up whether the G code is editable or not) 0: Disable 1: Enable	0~1			
		1	Source of calling macro file 0: CF card 1: INTER memory	0~1			
		3	Feed speed setting (To set the cutting speed by SF setting function key) 0: Disable 1: Enable	0~1			
		4	Program reset after edit (To set if the cursor returns to the program start line after editing the	0~1			

No.	Name	Description	Default	Range	Remark
		file.) 0: Disable 1: Enable			
10018	Background color	Background color	LIGHTGRAY	0~65535	
10019	Title color	Text color in the title bar	BLACK	0~65535	
10020	Mode color	Text color in the mode bar	DARKBLUE	0~65535	
10021	Function bar text color	Text color of the function bar	BLACK	0~65535	
10022	Label text font color	Label text font color	BLACK	0~65535	
10023	Numeric text color	Numeric text color	BLUE	0~65535	
10024	Grid line color	Grid line color	BLACK	0~65535	
10025	System cursor color	System cursor color	COLOR_S07	0~65535	
10026	System highlight text color	System highlight text color	WHITE	0~65535	
10027	Software panel cursor color	Software panel cursor color	YELLOW	0~65535	
10028	System alarm color	System alarm color	RED	0~65535	
10029	User alarm color	User alarm color	BLUE	0~65535	
10042	Highlighted text color of software panel	Highlighted text color of software panel	COLOR_S07	0~65535	
10043	Disable [PAR] group item	Bit | Description | Range 0 | [Operation parameter] screen display 0: Enable 1: Disable | 0~1 1 | [Tool magazine parameter] screen display 0: Enable 1: Disable | 0~1 2 | [Spindle parameter] screen display 0: Enable 1: Disable | 0~1 3 | [Mechanical parameter] screen display | 0~1	0	0~65535	P

No.	Name	Description			Default	Range	Remark
			0: Enable 1: Disable				
		4	[Home parameter] screen display 0: Enable 1: Disable	0~1			
		5	[Compensation parameter] screen display 0: Enable 1: Disable	0~1			
		6	[System parameter] screen display 0: Enable 1: Disable	0~1			
		7	[MLC parameter] screen display 0: Enable 1: Disable	0~1			
		8	[Graphic parameter] screen display 0: Enable 1: Disable	0~1			
		9	[Servo parameter] screen display 0: Enable 1: Disable	0~1			
		10	[Channel Setup] screen display 0: Enable 1: Disable	0~1			
		11	[RIO setting] screen display 0: Enable 1: Disable	0~1			
10044	Channel 0 teach setting	The teach function of each axis (When it is enabled, this parameter value is valid when teach function of [PRG] group is			0	0~65535	P

No.	Name	Description			Default	Range	Remark
		used in JOG or MPG mode.)					
		Bit	Description	Range			
		0	Teach of Axis X 0: Disable 1: Enable	0~1			
		1	Teach of Axis Y 0: Disable 1: Enable	0~1			
		2	Teach of Axis Z 0: Disable 1: Enable	0~1			
		3	Teach of Axis A 0: Disable 1: Enable	0~1			
		4	Teach of Axis B 0: Disable 1: Enable	0~1			
		5	Teach of Axis C 0: Disable 1: Enable	0~1			
		6	Teach of Axis U 0: Disable 1: Enable	0~1			
		7	Teach of Axis V 0: Disable 1: Enable	0~1			
		8	Teach of Axis W 0: Disable 1: Enable	0~1			
10045	Disable [DGN] group item	Bit	Description	Range	0	0~65535	P
		0	Display [Tuning] screen 0: Enable 1: Disable	0~1			
		1	Display [TEXT WR] function 0: Enable 1: Disable	0~1			
		2	Display [LOGO WR]	0~1			

No.	Name	Description	Default	Range	Remark
		0: Enable 1:Disable			

12.10 MLC setting

This function sets up the display environment of the component device and color of the MLC ladder diagram.

See the operation steps below:

1. Press the PAR key to enter the screen of [PARAMETER].
2. Press the ▶ key to switch to the screen with function bar.
3. Press the **MLC** to enter the MLC setup screen.
4. Use ↑ and ↓ keys to move the cursor to the desired data field and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.10.1.

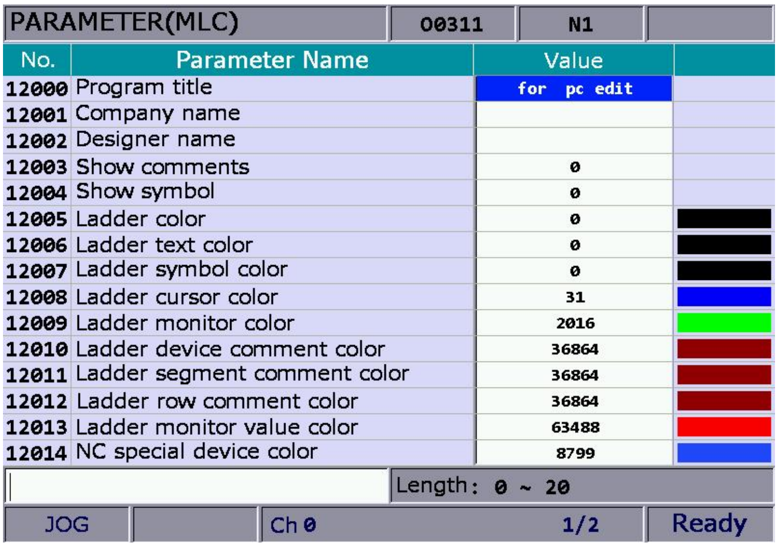


Figure 12.10.1

5. Press the ENTER key to complete the setting.
6. As for the setting of color item, press the **Color** and the color selection dialog box will pop up.
7. To reset the system environment back to its factory defaults status, press the **Default** and a confirmation dialog box will pop up.
8. Press “Y” (Yes) and the ENTER key to reset the system back to its factory defaults status.

12

12.10.1 MLC Parameter setting

No.	Item	Description	Default	Range	Remark
12000	Program title	Program title			
12001	Company name	Company name	0		
12002	Designer’s name	Designer’s name	0		
12003	Display remarks	Display remarks 0: off 1: on	0	0~1	
12004	Display symbols	Display symbols 0: off 1: on	0	0~1	
12005	Color of ladder diagram	Color of the ladder diagram	BLACK	0~65535	
12006	Text color of ladder diagram	Text color in the ladder diagram	BLACK	0~65535	
12007	Symbol color of ladder diagram	Symbol color in the ladder diagram	BLACK	0~65535	
12008	Cursor color of ladder diagram	Cursor color in the ladder diagram	LIGHT BLUE	0~65535	
12009	Monitoring color of ladder diagram	Monitoring color in the ladder diagram	LIGHT GREEN	0~65535	
12010	Device remark color in ladder diagram	Remark color for device in the ladder diagram	BROWN	0~65535	
12011	Remark color for sections in ladder diagram	Remark color for sections in the ladder diagram	BROWN	0~65535	
12012	Remark color for rows in ladder diagram	Remark color for rows in the ladder diagram	BROWN	0~65535	
12013	Color of monitoring value in ladder diagram	Color of monitoring value in the ladder diagram	LIGHTRED	0~65535	
12014	Color of special NC devices	Color of special NC devices	COLOR_S2B	0~65535	
12015	Color of special MLC devices	Color of special MLC devices	MAGENTA	0~65535	
12016	Enable MLC editing protection	Bit / Description / Ran-ge 0 / MLC editiong portection When this function is / 0~1	1	0~65535	

<table>
<tr><th>No.</th><th>Item</th><th colspan="3">Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td></td><td></td><td></td><td>enabled, editing the MLC program is only allowed in EDIT mode)
0: Disable
1: Enable</td><td></td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td>1</td><td>MLC display
0: Enable
1: Disable</td><td>0~1</td><td></td><td></td><td></td></tr>
<tr><td></td><td></td><td>2</td><td>MLC program auto backup
(Back up the MLC program in CF card when this function is enabled
0: Disable
1: Enable</td><td>0~1</td><td></td><td></td><td></td></tr>
<tr><td rowspan="3">12017</td><td rowspan="3">MLC Utility</td><td>Bit</td><td>Description</td><td>Range</td><td rowspan="3">0</td><td rowspan="3">0~65535</td><td rowspan="3"></td></tr>
<tr><td>0</td><td>Setting is valid once MLC program is loaded. (When this function is enabled, the setting is effective once the MLC is loaded)
0: Disable
1: Enable</td><td>0~1</td></tr>
<tr><td>1</td><td>MLC system record
0: Disable
1: Enable; it provides the opened filename, total machining time, single machining time
Current year & month
<table><tr><th colspan="2">D1107</th></tr><tr><td>Year-2000</td><td>month</td></tr></table>/day & hour
<table><tr><th>D1108</th></tr></table></td><td>0~1</td></tr>
</table>

12

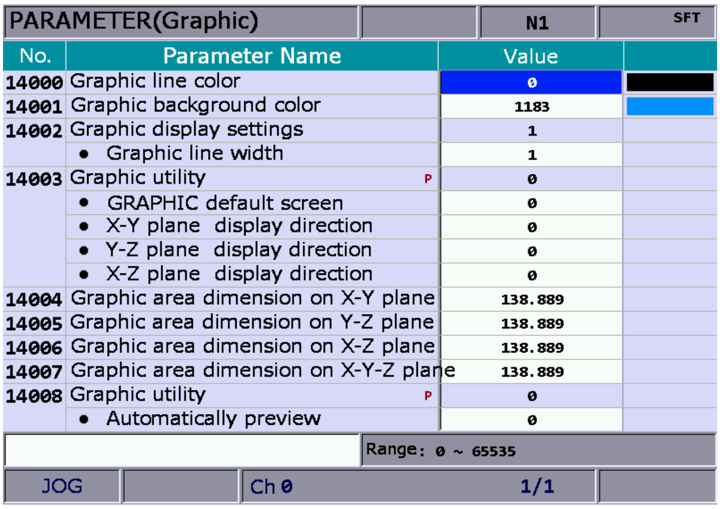
<table>
<tr><th>No.</th><th>Item</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td></td><td></td><td><table><tr><td>Day</td><td>hour</td></tr><tr><td colspan="2">/min & sec</td></tr><tr><td colspan="2">D1109</td></tr><tr><td>minute</td><td>Second</td></tr></table></td><td></td><td></td><td></td></tr>
<tr><td>12027</td><td>Enable user alarm 0</td><td><table>
<tr><th>Bit</th><th>Description</th><th>Range</th></tr>
<tr><td>0</td><td>Enable A0~A15 alarm trigger</td><td>0~1</td></tr>
<tr><td>1</td><td>Enable A16~A31 alarm trigger</td><td>0~1</td></tr>
<tr><td>2</td><td>Enable A32~A47 alarm trigger</td><td>0~1</td></tr>
<tr><td>3</td><td>Enable A48~A63 alarm trigger</td><td>0~1</td></tr>
<tr><td>4</td><td>Enable A64~A79 alarm trigger</td><td>0~1</td></tr>
<tr><td>5</td><td>Enable A80~A95 alarm trigger</td><td>0~1</td></tr>
<tr><td>6</td><td>Enable A96~A111 alarm trigger</td><td>0~1</td></tr>
<tr><td>7</td><td>Enable A112~A127 alarm trigger</td><td>0~1</td></tr>
<tr><td>8</td><td>Enable A128~A143 alarm trigger</td><td>0~1</td></tr>
<tr><td>9</td><td>Enable A144~A159 alarm trigger</td><td>0~1</td></tr>
<tr><td>10</td><td>Enable A160~A175 alarm trigger</td><td>0~1</td></tr>
<tr><td>11</td><td>Enable A176~A191 alarm trigger</td><td>0~1</td></tr>
<tr><td>12</td><td>Enable A192~A207 alarm trigger</td><td>0~1</td></tr>
<tr><td>13</td><td>Enable A208~A223 alarm trigger</td><td>0~1</td></tr>
<tr><td>14</td><td>Enable A224~A239 alarm trigger</td><td>0~1</td></tr>
<tr><td>15</td><td>Enable A240~A255 alarm trigger</td><td>0~1</td></tr>
</table></td><td>0</td><td>0~65535</td><td>P</td></tr>
</table>

<table>
<tr><th>No.</th><th>Item</th><th colspan="3">Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td rowspan="17">12028</td><td rowspan="17">Enable user alarm 1</td><td>Bit</td><td>Description</td><td>Range</td><td rowspan="17">0</td><td rowspan="17">0~65535</td><td rowspan="17">P</td></tr>
<tr><td>0</td><td>Enable A256~A271 alarm trigger</td><td>0~1</td></tr>
<tr><td>1</td><td>Enable A272~A287 alarm trigger</td><td>0~1</td></tr>
<tr><td>2</td><td>Enable A288~A303 alarm trigger</td><td>0~1</td></tr>
<tr><td>3</td><td>Enable A304~A319 alarm trigger</td><td>0~1</td></tr>
<tr><td>4</td><td>Enable A320~A335 alarm trigger</td><td>0~1</td></tr>
<tr><td>5</td><td>Enable A336~A351 alarm trigger</td><td>0~1</td></tr>
<tr><td>6</td><td>Enable A352~A367 alarm trigger</td><td>0~1</td></tr>
<tr><td>7</td><td>Enable A368~A383 alarm trigger</td><td>0~1</td></tr>
<tr><td>8</td><td>Enable A384~A399 alarm trigger</td><td>0~1</td></tr>
<tr><td>9</td><td>Enable A400~A415 alarm trigger</td><td>0~1</td></tr>
<tr><td>10</td><td>Enable A416~A431 alarm trigger</td><td>0~1</td></tr>
<tr><td>11</td><td>Enable A432~A447 alarm trigger</td><td>0~1</td></tr>
<tr><td>12</td><td>Enable A448~A463 alarm trigger</td><td>0~1</td></tr>
<tr><td>13</td><td>Enable A464~A479 alarm trigger</td><td>0~1</td></tr>
<tr><td>14</td><td>Enable A480~A495 alarm trigger</td><td>0~1</td></tr>
<tr><td>15</td><td>Enable A4960~A511 alarm trigger</td><td>0~1</td></tr>
</table>

12

12.11 Graph parameter

The graph parameter defines the display range of motion trails and provides plotting settings for GRA group.



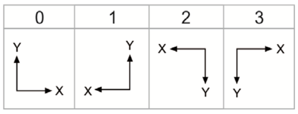
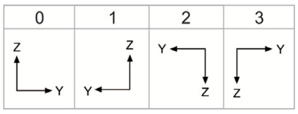
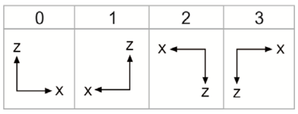
PARAMETER(Graphic)		N1	SFT
No.	Parameter Name	Value	
14000	Graphic line color	0	
14001	Graphic background color	1183	
14002	Graphic display settings	1	
	• Graphic line width	1	
14003	Graphic utility P	0	
	• GRAPHIC default screen	0	
	• X-Y plane display direction	0	
	• Y-Z plane display direction	0	
	• X-Z plane display direction	0	
14004	Graphic area dimension on X-Y plane	138.889	
14005	Graphic area dimension on Y-Z plane	138.889	
14006	Graphic area dimension on X-Z plane	138.889	
14007	Graphic area dimension on X-Y-Z plane	138.889	
14008	Graphic utility P	0	
	• Automatically preview	0	
	Range: 0 ~ 65535		
JOG		Ch 0	1/1

Figure 12.11.1

See the operation steps below:

1. Press the PAR key to enter the screen of [PARAMETER].
2. Press the ▶ key to switch to the screen with function bar.
3. Press the **Graphic** to enter the graph parameter setup screen.
4. Use ↑ and ↓ keys to move the cursor to the desired data field and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen) as shown in Figure 12.11.1.
5. Press the ENTER key to complete the setting.
6. As for the setting of color item, press the **Color** and the color selection dialog box will pop up.
7. To reset the system environment back to its factory defaults status, press the **Default** and a confirmation dialog box will pop up.
8. Press “Y” (Yes) and the ENTER key to reset the system back to its factory defaults status.

12.11.1 Graphic parameter setting

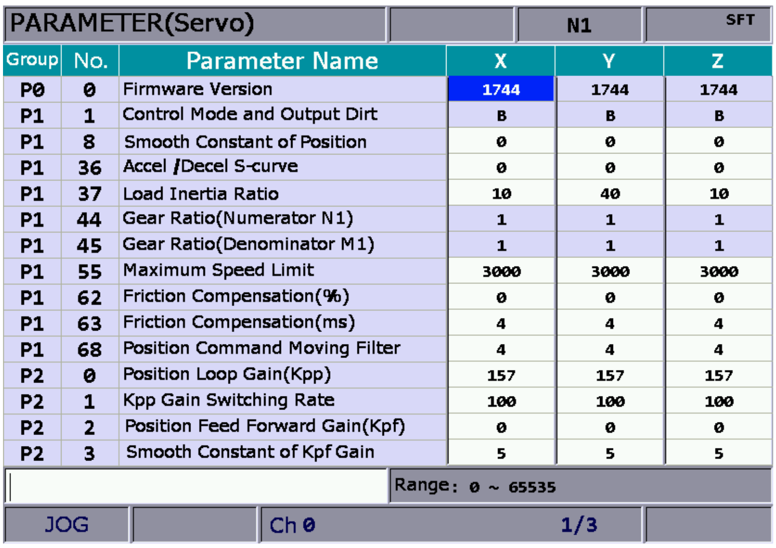
<table>
<tr><th>No.</th><th>Item</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td>14000</td><td>Line color</td><td>Line color of the graph</td><td>BLACK</td><td>0~65535</td><td></td></tr>
<tr><td>14001</td><td>Background color</td><td>Background color of the graph</td><td>SEA</td><td>0~65535</td><td></td></tr>
<tr><td>14002</td><td>Line width</td><td>Set the graph display
<table>
<tr><th>Bit</th><th>Description</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>Workpiece utility</td><td>
<table>
<tr><th>Bit</th><th>Description</th><th>Range</th></tr>
<tr><td>0</td><td>Default graphic display</td><td>0~1</td></tr>
<tr><td>1</td><td>Orientation of X-Y plane
</td><td>0~3</td></tr>
<tr><td>3</td><td>Orientation of Y-Z plane
</td><td>0~3</td></tr>
<tr><td>5</td><td>Orientation of X-Z plane
</td><td>0~3</td></tr>
</table></td><td>0</td><td>0~65535</td><td>P</td></tr>
<tr><td>14004</td><td>Graphic dimension on X-Y plane</td><td>Graphic dimension on X-Y plane
Unit: mm</td><td>200</td><td>5~ 100000</td><td></td></tr>
<tr><td>14005</td><td>Graphic dimension on Y-Z plane</td><td>Graphic dimension on Y-Z plane
Unit: mm</td><td>200</td><td>5~ 100000</td><td></td></tr>
<tr><td>14006</td><td>Graphic dimension on X-Z plane</td><td>Graphic dimension on X-Z plane
Unit: mm</td><td>200</td><td>5~ 100000</td><td></td></tr>
<tr><td>14007</td><td>Graphic dimension on X-Y-Z plane</td><td>Graphic dimension on X-Y-Z plane
Unit: mm</td><td>200</td><td>5~ 100000</td><td></td></tr>
</table>

No.	Item	Description			Default	Range	Remark
		Bit	Description	Range			
14008	Workpiece utility	0	Auto look ahead 0: Off 1: On	0~1	0	0~65535	
		1	Keep the graph after M30 is executed (When this function is enabled, the graph will be kept after M30 is executed.) 0: Disable 1: Enable	0~1			

12.12 Servo parameter

Through the servo parameter setup screen, the servo end can control and set up parameters. See the operation steps below:

1. Press the PAR key to enter the screen of [PARAMETER].
2. Press the ▶ key to switch to the screen with function bar.
3. Press the **Servo** to enter the servo parameter setup screen.
4. Use ↑ and ↓ keys to move the cursor to the desired data field and enter the proper values (Refer to the recommended values displayed at lower right corner of the screen).
5. Press ENTER key to complete the setting.



Group	No.	Parameter Name	X	Y	Z
P0	0	Firmware Version	1744	1744	1744
P1	1	Control Mode and Output Dirt	B	B	B
P1	8	Smooth Constant of Position	0	0	0
P1	36	Accel /Decel S-curve	0	0	0
P1	37	Load Inertia Ratio	10	40	10
P1	44	Gear Ratio(Numerator N1)	1	1	1
P1	45	Gear Ratio(Denominator M1)	1	1	1
P1	55	Maximum Speed Limit	3000	3000	3000
P1	62	Friction Compensation(%)	0	0	0
P1	63	Friction Compensation(ms)	4	4	4
P1	68	Position Command Moving Filter	4	4	4
P2	0	Position Loop Gain(Kpp)	157	157	157
P2	1	Kpp Gain Switching Rate	100	100	100
P2	2	Position Feed Forward Gain(Kpf)	0	0	0
P2	3	Smooth Constant of Kpf Gain	5	5	5

Figure 12.12.1

12.12.1 Servo parameter setting

Group	No.	Name	Description	Default	Range	Remark
P0	0	Firmware version	This parameter shows the firmware version of the servo drive.		0	
P1	1	Input setting of control mode and control command	Setting for different control modes Bit | Description | Range 8 | Direction of Torque output | 0~1	0	0x00~ 0x110F (HEX)	
P1	8	Smooth constant of position command (Low-pass Filter)	Smooth constant of position command Unit: 10 msec	0	0~0	Range for CNC
P1	36	Acceleration / Deceleration constant of S-Curve	Acceleration and deceleration constant of S-Curve Unit: msec	0	0~0	Range for CNC
P1	37	Inertia ratio and load weight ratio to servo motor	The inertia ratio and load weight ratio to the servo motor. Unit: 0.1 times	10	0~2000	
P1	44	Gear ratio (Numerator) (N1)	Set up the numerator of multiple gear ratio Unit: pulse	1	$1\sim(2^{29}-1)$	Read only
P1	45	Gear ratio (Denominator) (M1)	Set up the denominator of gear ratio (M) Unit: pulse	1	$1\sim(2^{31}-1)$	Read only
P1	55	Maximum speed limit	Set up the maximum speed of the servo motor. The defalt value is set to the rated speed Unit: r/min	0	0~65535	
P1	62	Friction compensation (%)	Set up the level of friction compensation Unit: %	0	0~100	
P1	63	Friction compensation (ms)	Set up the smoothing constant of friction compensation. Unit: ms	4	4~4	Range for CNC

Group	No.	Name	Description	Default	Range	Remark
P1	68	Position command (moving filter)	Set up the position command (moving filter) Unit: ms	4	0~100	
P2	0	Position loop gain	Set up the position loop gain Unit: rad/s	35	0~2047	
P2	1	Switching rate of position loop gain	Set up the changing rate of position loop gain. Unit: %	100	10~500	
P2	2	Position feed forward gain	Set up the position feed forward gain Unit: %	50	0~100	
P2	3	Smooth constant of position feed forward gain	Set up the smooth constant of position feed forward gain Unit: msec	5	2~100	
P2	4	Speed loop gain	Set up the speed loop gain Unit: rad/s	500	0~8191	
P2	5	Switching rate of speed loop gain	Set up the changing rate of speed loop gain Unit: %	100	10~500	
P2	6	Speed integral compensation	Speed integral compensation Unit: rad/s	100	0~1023	
P2	7	Speed feed forward gain	Speed feed forward gain Unit: %	0	0~100	
P2	9	DI debouncing time	Set up the DI debouncing time Unit: 2 msec	2	0~20	
P2	23	Resonance suppression (Notch filter) (1)	Set up the first group of resonance suppression. Unit: Hz	1000	50~1000	
P2	24	Resonance suppression attenuation rate (1)	The first resonance suppression (notch filter) attenuation rate. When this parameter is set to 0, the function of Notch filter is disabled. Unit: dB	0	0~32	
P2	25	Low-pass filter of resonance suppression	Set the low-pass filter of resonance suppression. When the value is set to 0, the function of low-pass filter is disabled. Unit: 0.1 msec	2	0~1000	
P2	26	Anti-interference gain	The gain against external interference Unit: 0.001	0	0~0	Range for CNC

Group	No.	Name	Description	Default	Range	Remark
P2	27	Gain switching and switching selection	Selection for gain switching condition and method	0	0~4 (HEX)	
P2	28	Gain switching time constant	Gain switching time constant Unit: 10 msec	10	0~1000	
P2	43	Resonance suppression (Notch filter) (2)	Set up the second group of resonance suppression. Unit: Hz	1000	50~2000	
P2	44	Resonance suppression (Notch filter) attenuation rate (2)	Set up the second group of resonance suppression (notch filter) attenuation rate. When it is set to 0, the function of notch filter is disabled. Unit: dB	0	0~32	
P2	45	Resonance suppression (Notch filter) (3)	Set up the third group of resonance suppression Unit: Hz	1000	50~2000	
P2	46	Resonance suppression (Notch filter) Attenuation Rate (3)	Set up the third group of resonance suppression (notch filter) attenuation rate. When it is set to 0, the function of notch filter is disabled. Unit: dB	0	0~32	
P2	47	Auto resonance suppression mode setting	0: fixed 1: auto fix after suppression 2: continuous auto suppression	1	0~2	
P2	49	Speed detection filter	Set up the speed detection filter Unit: sec	0	0~1F	
P2	53	Position integral compensation	The greater the position integral is, the smaller steady-sate error will be. Unit: rad/s	0	0~1023	
P2	69	Absolute encoder setting	Set up the motor type 0: Incremental type 1: Absolute type	0	0~1	Servo Power on
P4	0	Fault record (N)	The latest fault record	0		Read only
P4	1	Fault record (N-1)	The last second fault record	0		Read only
P4	2	Fault record (N-2)	The last third fault record	0		Read only

12

Group	No.	Name	Description	Default	Range	Remark
P4	3	Fault record (N-3)	The last fourth fault record	0		Read only
P4	4	Fault record (N-4)	The last fifth fault record	0		Read only
P5	0	Firmware sub-version	Firmware sub-version of the servo drive	0		Read only

12.13 Channel setting (Ch/Axis Setting)

This function sets up the number and definition of the axis employed by a system as shown in Figure 12.13.1. The system mode cannot be set up in Auto and Manual modes.

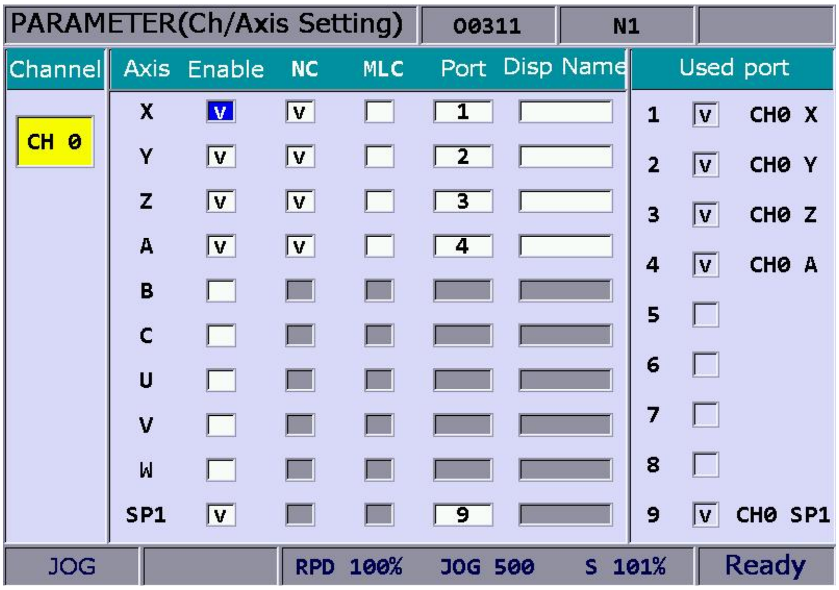


Figure 12.13.1

See the operation steps below:

1. Press the PAR key to enter the screen of [PARAMETER].
2. Repeatedly press the ▶ key to move the cursor to the third row of the function page in this layer.
3. Press the **Config** to enter the screen for channel setup.
4. Use ↑ and ↓ keys to move the cursor to the desired axis function fields, and then press the ENTER key to select the axis and enable its attribute setting. Users can select the axis attribute of either NC or MLC as desired.
5. To define axis attributes: Use ↑, ↓, ←, and → keys to move the cursor to the desired field and press the ENTER key to check the field.
6. After the axis attribute is defined, use ↑, ↓, ←, and → keys to move the cursor to the port field of the axis, press the ENTER key and the port number entry box for setting up the port number will pop up, Then, press 1~ 9 keys to enter a unique port number, press the ENTER key and the port number of the axis is set.
7. Press the **OK** after all axes are defined.
8. Restart the control system to validate the setting.

Note:
1. Please check to activate the axis name before enabling the axis. Users can set up the definition to control the axis only after it is activated. Select either the NC or MLC axis (not both), and assign a port number (unique from other axis port numbers).
2. To disable (cancel) the axis function, move the cursor to the specified field and press the ENTER key to uncheck the item. Then the function of this axis will be disabled.
3. Parameter fields marked with the letter 'P' indicates that changes can take effect only after the NC control system is restarted. Otherwise, changes take effect immediately.

12

12.14 RIO setting

The NC numerical control system can have external control switch by increasing the I/O expansion module. The I/O function module can be enabled in RIO setting page as shown in Figure 12.14.1. See the operation steps below:

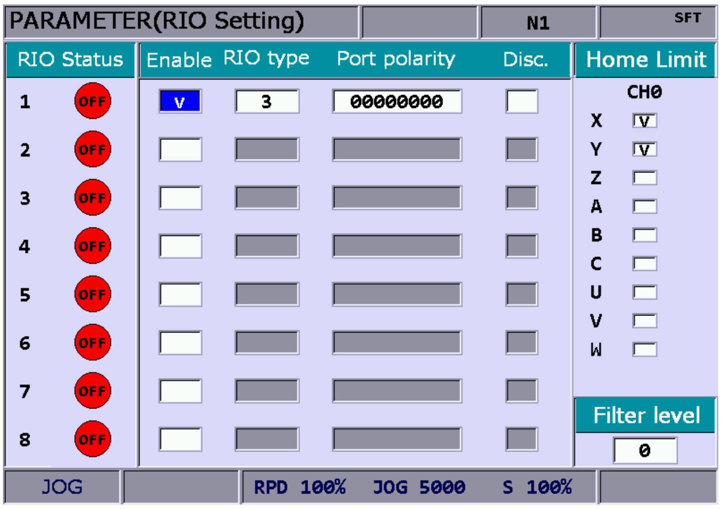


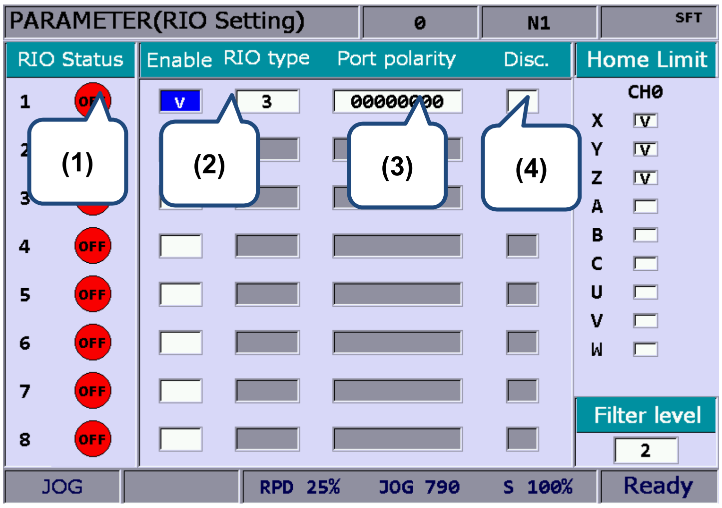
Figure 12.14.1

1. Press the **PAR** key to enter the screen of [PARAMETER].
2. Repeatedly press the ▶ key to move the cursor to the third row of the function page in this layer.
3. Press the **Set RIO** to enter the I/O expansion module setup page.
4. Use ↑ and ↓ keys to move the cursor to the desired RIO port fields, press the **ENTER** key to check the field and display the I/O setup screen of the selected port.
5. Use ← and → keys to move the cursor to the polarity setup field, press the **ENTER** key and the entry box will pop up. Then, press the **ENTER** key after data entry and the polarity is set.
6. Then, use ← and → keys to move to the field of [Disc.] (output when disconnected), press the **ENTER** key to enable/disable the setting.
7. Press the **OK** after all I/O modules function are set.

12

12.14.1 RIO setting detail

RIO setting: Click on **OK** after setting is completed.



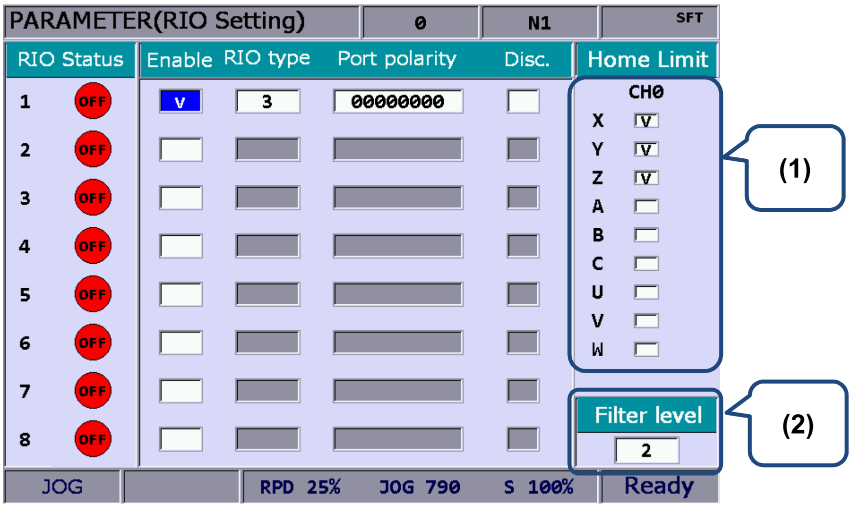
(1) Use the RIO station number to show the connection status, M2832 is station No.0.

(2) Select the station number. Set RIO type to 3 as DIO and the setting of the others is as the follows. AD/DA(set 0); DA (set 1); AD (set 2)

(3) The setting of DI port polarity, which only can be set to station No.0 and can set DI0~DI31 (32 points in total).

(4) Check this item means DO is on when disconnection.

Polarity setting of the origin: Click on **OK** after setting is completed.



(1) Axis selection. The positive, negative and origin DI of Axis X ~ A are input by port AXIS1~4 of the controller. According to the axis number and name selected, starting from X256, each axis occupies three DI points, positive, negative and origin respectively.

For example, if axis Y and Z are selected, then:
Y-axis: positive DI (X256), negative DI (X257) and origin (X258); Z-axis: positive DI (X259), negative DI (X260) and origin (X261)

When DI is on, the special M remains unchanged.

X positive limit M2144	X negative limit M2145	X origin signal M2146
Y positive limit M2148	Y negative limit M2149	Y origin signal M2150
Z positive limit M2152	Z negative limit M2153	Z origin signal M2154

(2) Set the DI filter level of RIO, each level is 40 micro seconds (10^-6 sec) There are 5 levels in total. All the DIs in RIO are applicable.

12.14.2 DA module setting (NC-EIO-DAC04)

12

The following is the steps to set DA module:

1. Go to the **RIO setting** screen > enable port 5 > select type 1. Then, D1464 ~ D1467 corresponds to the output points 0 ~3 respectively.
2. Switch the node number to 4 on the DA card.
3. The connection for communication is the same as that of Remote IO.
4. Once completing the above steps, please restart the system. Now, if you set D1464 to 1024, you can find 1.25 V at the output point 0 on the DA card. (+/-10V corresponds to 8191 ~ -8192)

See the following table for the corresponding port number for MLC special D in the **RIO setting** screen:

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

As NC systems has many types of parameter, this function enables users to search and point to the desired parameter fields by entering parameter codes. It is faster and easier to access to the screen containing the given parameter.

See the operation steps below:

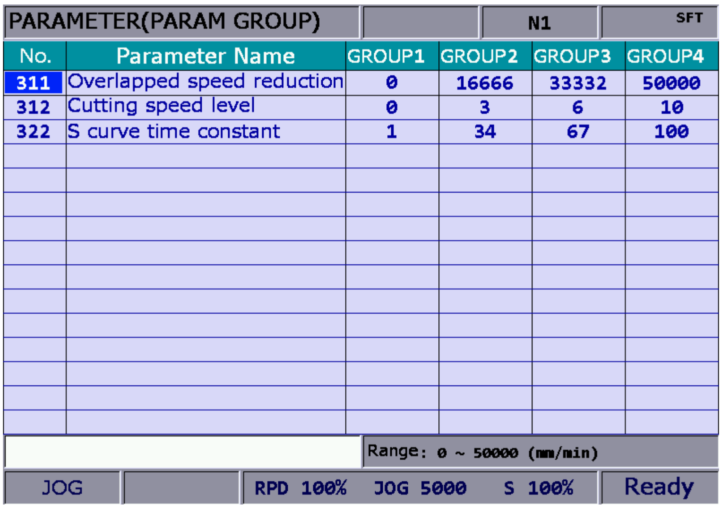
1. Press the **PAR** key to enter the screen of [PARAMETER].
2. Repeatedly press the ▶ key to move the cursor to the third row of the function page in this layer.
3. Enter the parameter code to be searched in the field located at the lower bottom of the screen.
4. Press the **Search** to start searching.

Note: Apart from using the function key to search the parameter, users can enter the parameter number in the screen of PAR group. The method is: **S** + **parameter number** and then press the ENTER key.

12.16 Paramter group (PARAM GROUP)

12

The NC numerical control system provides many types of parameters. Users can define the customized parameter group function and select the appropriate parameter groups according to different demands and applications. See Figure 12.16.1 below.



PARAMETER(PARAM GROUP) | N1 | SFT

No.	Parameter Name	GROUP1	GROUP2	GROUP3	GROUP4
311	Overlapped speed reduction	0	16666	33332	50000
312	Cutting speed level	0	3	6	10
322	S curve time constant	1	34	67	100

Range: 0 ~ 50000 (mm/min)

JOG | RPD 100% JOG 5000 S 100% | Ready

Figure 12.16.1

See the operation steps below:

1. Press the PAR key to enter the screen of [PARAMETER].
2. Repeatedly press the ▶ key to move the cursor to the third row of the function page in this layer.
3. Press the **PAR GROUP** to enter the parameter group setup page.
4. Enter the specified parameter number in the field of [Number] and press the ENTER key. Then, parameter name will appear on the screen.
5. Press the **PAR SEQUENCE** and the parameter number on this setting page will show in sequence. Press the **PAR SEQUENCE** to display the parameter number from small > big. Press the **PAR SEQUENCE** again, the parameter number will be displayed from big to small.
6. Use ⇦ and ⇨ keys to move the cursor to the desired group field and press the ENTER key to complete the setting of parameter group. Or press the **READ PAR** when the cursor stops at the field of [GROUP], a confirmation box of reading the parameter will pop up. Then, press “Y” (Yes) and press the ENTER key again to read the parameter value.
7. If you wish to delete the parameter group, use ⇦ and ⇨ keys to move the cursor to the desired group field, press the **DEL GROUP** and a confirmation box of “Delete the group” will pop up. Press “Y” and the ENTER key to delete the group.
8. When entering multiple parameter groups, press the **ALLOCATE** and a confirmation box will pop up. Press “Y” and the ENTER key. The system will divide the range of parameter value by the group number. Then the parameter value will be allocated to each group that you currently applied.
9. After completing the setting of parameter group, press the **SAVE** to see the setting screen and result and a confirmation box will pop. Then, press “Y” (Yes) and press the ENTER key again to save the setting.
10. Use ⇦ and ⇨ keys to move the cursor to the specified group field and press the **WRITE PAR**. A confirmation box will then pop up. Press “Y” and the ENTER key again to

write the value to the corresponding parameters.

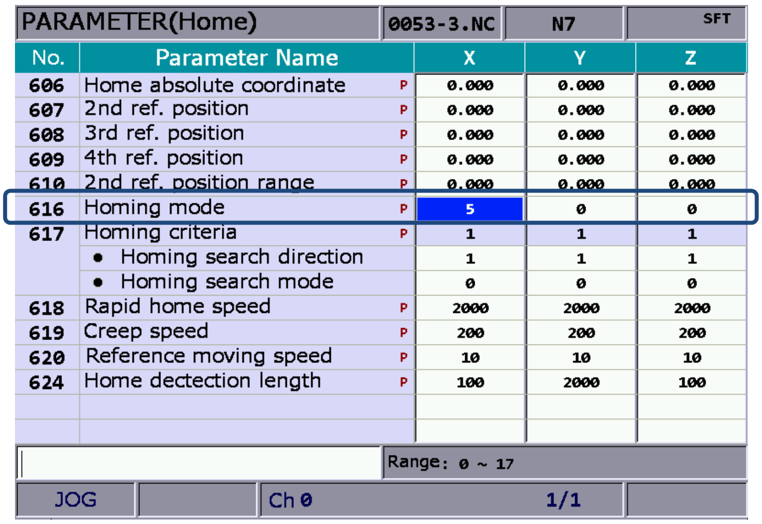
Note:
1. Writing the value to parameters will replace the original parameter value in the system. Please make sure the parameter value is correct beforehand.
2. Up to 20 groups with maximum 20 parameters for each group are supported.

12.17 Other special setting

12.17.1 Setting for absolute motors

When using absolute type motor with NC system, please follow the steps below to set up the system:

1. Go to the Origin parameter (Home) setting screen. Set parameter 616 (Homing mode) to 5 (incremental type and absolute type motors can be used at the same time. When an absolute motor is installed for the first time, please re-power on the servo and the controller once the setting is complete. See the figure below.



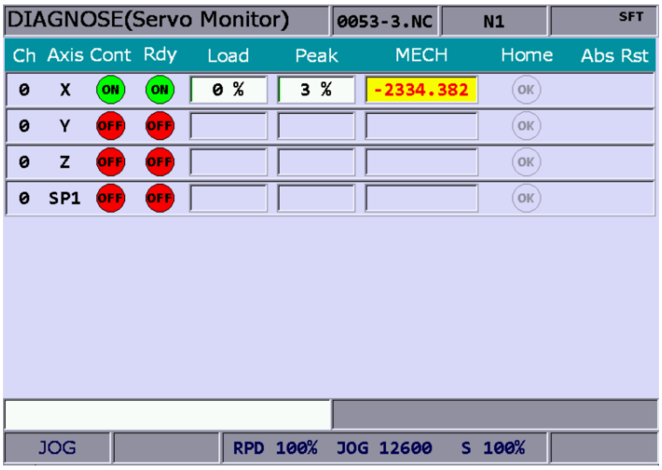
PARAMETER(Home) | 0053-3.NC | N7 | SFT

No.	Parameter Name		X	Y	Z
606	Home absolute coordinate	P	0.000	0.000	0.000
607	2nd ref. position	P	0.000	0.000	0.000
608	3rd ref. position	P	0.000	0.000	0.000
609	4th ref. position	P	0.000	0.000	0.000
610	2nd ref. position range	P	0.000	0.000	0.000
616	Homing mode	P	5	0	0
617	Homing criteria	P	1	1	1
	• Homing search direction		1	1	1
	• Homing search mode		0	0	0
618	Rapid home speed	P	2000	2000	2000
619	Creep speed	P	200	200	200
620	Reference moving speed	P	10	10	10
624	Home dectection length	P	100	2000	100

Range: 0 ~ 17

JOG | Ch 0 | 1/1

2. Once the parameter is set, execute homing procedure of absolute motors by going to screen of [DGN] > [System monitoring] > [Servo monitoring]. See the figure below.



3. To exeute homing procedure, the system setting has to be done in **JOG** or **MPG** mode. Firstly, change the mechanical coordinate to the proper position. Then, enter **1** and press **ENTER**, the homing is complete. Meanwhile, the origin indicator will light up and mechanical coordinates displays 0; it means this axis has completed the homing procedure.

Note: When it is in MPG mode, setting is valid only for the specified MPG axis. For instance, if axis X of MPG is selected, complete the setting by entering **1** and pressing **ENTER**. If a servo alarm occurs, this homing flag will be off. The alarms that will affect the homing procedures are:
AL060: The absolute position is lost, homing procedure is needed.
AL061: Battery undervoltage, please change the battery.
AL069: Wrong motor type, please make sure the absolute type encoder is connected.

12.17.2 Synchronous motion function setting

Application Description: Axis A (Slave) follows axis Z (Master) and the following direction is the same. Assume that M13 is set to “synchronous motion enable” and M14 is “synchromous motion disable”. The parameter is set as follows.

1. Set parameter 350 to 13;
2. Set parameter 351 to 14;
3. Set parameter 364 (Synchronous control A) to 3.

When M13 is executed, MLC will trigger M1088 (Synchronous control flag) and M1092 (Slave A follows the Master). When these flags are on, the synchronous control of axis A will be enabled as soon as axis Z starts moving. If G01A10 is executed, an alarm will occur. To stop the synchronous control, execute M14 and the M1088 and M1092 will be off and the control will be canceled.

When synchronous control is in effect, it is effective in Auto and MDI mode as well as JOG and Home mode. The synchronous control will not be canceled until M14 is executed (synchromous motion disable) (Exception: When axis A follows axis Z during tapping, axis A’s motion is not synchronous with axis Z. See the following notes:

(1) When the Master is set, this axis cannot be specified as a Slave.
(2) When the Slave is set, this axis cannot be specified as a Master.
(3) Mutiple slaves can follow the same Master.
(4) When homing to the origin under synchronous control, the slave still follows the master.
(5) Press **RESET** won’t cancel the synchronous motion.

Program:

```
G54X0Y0Z0A0
G90G54G0X10.Y10.Z10.
Z50.
A0
M13
Z0.
Z111.
G4X2.
Z150.
M14
A100.
A51.
M30
```

Relevant parameter setting:

<table>
<tr><th>No.</th><th>Name</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td rowspan="10">350
~
359</td><td rowspan="10">Halt M code
1 ~10</td><td>Halt M code 1 (0: no setting)</td><td rowspan="10">0</td><td rowspan="10">0~1000</td><td rowspan="10">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>Halt M code 9</td></tr>
<tr><td>Halt M code 10</td></tr>
<tr><td>360</td><td>Synchronous control direction</td><td>Synchronous control direction
Bit0 ~ 5: Synchronous control Axis X ~ C
0: Same direction
1: Reverse direction
<table>
<tr><th>Bit</th><th>Description</th><th>Range</th></tr>
<tr><td>0</td><td>Synchronous control direction X</td><td>0~1</td></tr>
<tr><td>1</td><td>Synchronous control direction Y</td><td>0~1</td></tr>
<tr><td>2</td><td>Synchronous control direction Z</td><td>0~1</td></tr>
<tr><td>3</td><td>Synchronous control direction A</td><td>0~1</td></tr>
<tr><td>4</td><td>Synchronous control direction B</td><td>0~1</td></tr>
<tr><td>5</td><td>Synchronous control direction C</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>Specify axis X as the Slave and set the Master to be followed. For example, to follow axis Y for synchronous control, set this parameter to 2.
0: Disable
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>Specify axis Y as the Slave and set the Master to be followed.
0: Disable
1 ~ 6: X ~ C</td><td>0</td><td>0~6</td><td>P</td></tr>
</table>

No.	Name	Description	Default	Range	Remark
363	Synchronous control Z	Specify axis Z as the Slave and set the Master to be followed. 0: Disable 1 ~ 6: X ~ C	0	0~6	P
364	Synchronous control A	Specify axis A as the Slave and set the Master to be followed. 0: Disable 1 ~ 6: X ~ C	0	0~6	P
365	Synchronous control B	Specify axis B as the Slave and set the Master to be followed. 0: Disable 1 ~ 6: X ~ C	0	0~6	P
366	Synchronous control C	Specify axis B as the Slave and set the Master to be followed. 0: Disable 1 ~ 6: X ~ C	0	0~6	P

Convert MLC command into NC command:

Function	Register No.	Function	Register No.
Trigger flag of synchronous control	M1088	The slave axis A follows the master axis	M1092
The slave axis X follows the master axis	M1089	The slave axis B follows the master axis	M1093
The slave axis Y follows the master axis	M1090	The slave axis C follows the master axis	M1094
The slave axis Z follows the master axis	M1091	-	-

12.17.3 Command transit setting

Application description: Transit command of axis Z (G01Z10.) to axis A (transition axis). Assume that M20 is set to “transit enable” and M21 is “transit disable”. The parameter setting is as follows:

1. Set parameter 350 to 20
2. Set parameter 351 to 21
3. Set parameter 374 (transfer control A) to 3

Execute M20 to trigger M1098 (Trigger flag of transit command controls) and M1102 (Axis A receives command from the master) of MLC program. When both flags are on, the command of axis Z will be transited to axis A (Axis Z remains unmoved). If G01A10 is executed, an alarm will occur. To stop the transit control, execute M21 to disable the command transit function and the special M flags will be off. This function can only be enabled (M20) and disabled (M21) in Auto and MDI mode. Please disable the transit control (M21) when program execution is complete. It cannot be used in JOG, MPG, or Home mode.

See the following notes:

1. When you have specified an axis as a transition axis that receives commands, do not specify this axis as a master axis.
2. When you have specified an axis as a master axis, do not set this axis to the transition axis.
3. A master can transit its command to multiple transition axes.
4. Transit function is not supported by Home mode.
5. Pressing **RESET** will not cancel the transit function.
6. When command of axis Z is transited to axis A, tool length compensation is supported.
7. Transiting turning cycle command for a mast Z is supported.

Program:

```
G54X0Y0Z0A0
G90G54G0X10.Y10.Z10.
Z50.
A0
M20   (Look ahead to M20 and enable transit control)
Z0.   (Coordinate display of axis Z is changing but actually it is axis A that is moving.)
Z111.
G4X2.
Z150.
M21   (Look ahead to M20 and disable transit control)
A100.
A51.
M30
```

Relevant parameter setting:

<table>
<tr><th>No.</th><th>Name</th><th>Description</th><th>Default</th><th>Range</th><th>Remark</th></tr>
<tr><td rowspan="10">350~359</td><td rowspan="10">Halt M code
1 ~10</td><td>Halt M code 1 (0: no setting)</td><td rowspan="10">0</td><td rowspan="10">0~1000</td><td rowspan="10">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>Halt M code 9</td></tr>
<tr><td>Halt M code 10</td></tr>
<tr><td>370</td><td>Transfer control
direction</td><td>Transfer control direction
Bit0 ~ 5: Synchronous control Axis X ~ C
0: same direction
1: reverse direction
<table>
<tr><th>Bit</th><th>Description</th><th>Range</th></tr>
<tr><td>0</td><td>Transfer control direction X</td><td>0~1</td></tr>
<tr><td>1</td><td>Transfer control direction Y</td><td>0~1</td></tr>
<tr><td>2</td><td>Transfer control direction Z</td><td>0~1</td></tr>
<tr><td>3</td><td>Transfer control direction A</td><td>0~1</td></tr>
<tr><td>4</td><td>Transfer control direction B</td><td>0~1</td></tr>
<tr><td>5</td><td>Transfer control 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>The command is transferred to axis X and the original command axis does not move. For example, to transfer the command of axis Y to axis X, set this parameter to 2.
0: Disable
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>The command is transferred to axis Y and the original command axis does not move. Axis Y will carry out the command.
0: Disable
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>The command is transferred to axis Z and the original command axis does not move. Axis Z will carry out the command.
0: Disable</td><td>0</td><td>0~6</td><td>P</td></tr>
</table>

12

No.	Name	Description	Default	Range	Remark
		1 ~ 6: X ~ C			
374	Transfer control A	The command is transferred to axis A and the original command axis does not move. Axis A will carry out the command. 0: Disable 1 ~ 6: X ~ C	0	0~6	P
375	Transfer control B	The command is transferred to axis B and the original command axis does not move. Axis B will carry out the command. 0: Disable 1 ~ 6: X ~ C	0	0~6	P
376	Transfer control C	The command is transferred to axis C and the original command axis does not move. Axis C will carry out the command. 0: Disable 1 ~ 6: X ~ C	0	0~6	P

Convert MLC command to NC command:

Function	Register No.	Function	Register No.
Trigger flag of transfer command controls	M1098	Axis A receives command from master axis	M1002
Axis X receives command from master axis	M1099	Axis B receives command from master axis	M1003
Axis Y receives command from master axis	M1000	Axis C receives command from master axis	M1004
Axis Z receives command from master axis	M1001	-	-

Convert NC command to MLC command:

Function	Register No.	Function	Register No.
Transfer function in execution	M2228	-	-

Software control panel (SOFT) Group

13

The SOFT group provides customized interface via CNC soft software. You can find the interface example in this chapter.

SOFT group is a special control function provided by NC numerical controller to replace the physical secondary control panel or special operation functions. With the CNC SOFT software, users can add a secondary control panel screen and use it to do exactly the operations available in the physical secondary control panel. This function can be used in environments without physical secondary control panel to support special repair servicing needs. Users may use it to add software keys with self-developed special controls for expanded functionality. This group function can replace the physical secondary control panel's control or function options.

Note: Here we use $\boxed{\text{Framed Text}}$ to indicate the keys in primary control panel. And **boldface letter** is used for indicating the function key.

13.1 Control panel

- Without physical control panel

See Figure 13.1.1 for an example of the function devices offered by this function. Keys and buttons of a physical secondary control panel are simulated with control components. Each device is turned on or off with relevant function key. Device types and priorities vary with user preference. Icons are sorted from bottom to top.

See the operation steps below for the operation of the control panel:

1. Press the $\boxed{\text{SOFT}}$ key to enter the SOFT group screen.
2. Press the **control panel** to enter the device function bar screen.
3. Use and keys to access the corresponding device function page as shown in Figure 13.1.1.

Figure 13.1.1

4. Use function keys to enable or disable device function.

■ With physical control panel

The software panel can define additional functions and locations which are required by expanded requirements. In a machine with physical secondary control panel, use the CNC SOFT software to add auxiliary configuration functions, including spindle tool release, auto chip removal and auto power off in the screen as shown in Figure 13.1.2.

Figure 13.1.2

See the operation steps described below for the operation of the control panel:

1. Press the **SOFT** key to enter the SOFT group screen.
2. Press the **control panel** to enter the device function bar screen.
3. Use ⬆ and ⬇ keys to enter the corresponding device function page.
4. Use function keys to enable or disable device function.

13

13.2 Factor regulation

Available factors are: cut feeding rate, fast feed rate, spindle speed, jog, and MPG.

Use the up and down arrow keys to select the factor type and operation as shown in Figure 13.2.1.

Range of cut feeding rate: 0% ~ 150% (in steps of 10%).

Range of fast feeding rate: F0, 25, 50, 100(%).

Range of spindle speed: 50% ~ 120% (in steps of 10%).

Range of jog factor: 0, 2, 3, 5, 8, 13, 20, 32, 50, 80, 120, 200, 320, 500, 790, 1260 mm/min.

Range of MPG factor: 0.001, 0.01, 0.1 (mm)

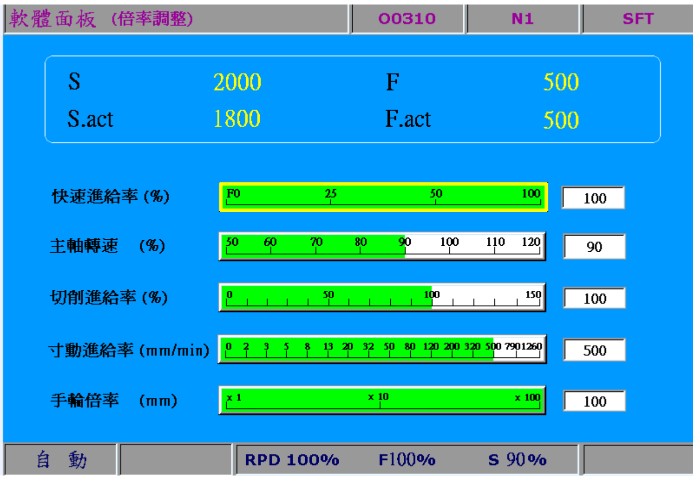


Figure 13.2.1

See the operation steps described below for factor regulation:

1. Press the **SOFT** key to enter the SOFT group screen.
2. Press the **factor regulation** to enter the factor regulation setup screen.
3. Use  and  keys to point the setup box to the specified regulation icon as shown in Figure 13.2.1.
4. Available options in the setup box are: increasing, decreasing, 100%, and 0%. Press the relevant function key to adjust factors as desired.

13

13.3 Axis operation

■ Without physical control panel

Use the SOFT group function to set the machine's individual axis for axial movements through software panel as shown in Figure 13.3.1.

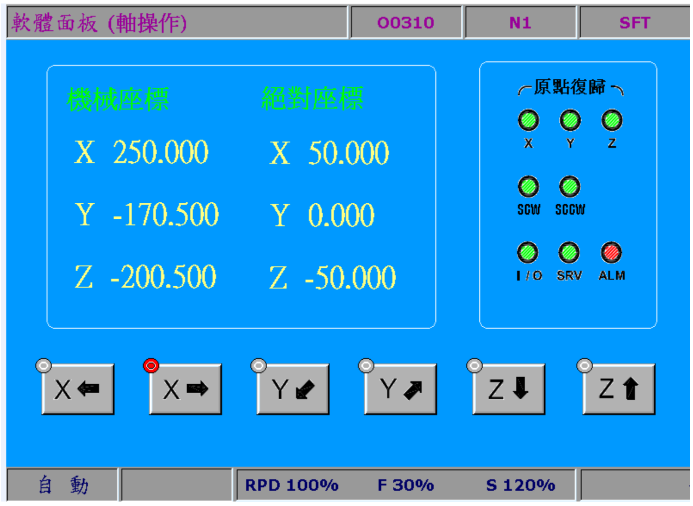


Figure 13.3.1

See the operation steps described below for axis operation:

1. Press the $\boxed{\text{SOFT}}$ key to enter the SOFT group function screen.
2. Press the **axis operation** to enter the axis operation screen.
3. Use function keys to do axial movements.

Note:

1. Configuration illustrations described here prioritize functions that are more likely to be used during machining. For instance, program execution, execution stops, and single block execution are placed in the first row. (They can be arranged as desired by the software.)
2. The travel distance (or speed) of axis operation varies with factor settings as described in Section 13.2.

(This page is intentionally left blank.)

MLC Special M and D Command

14

This chapter provides detailed introduction to the special controlling devices of NC series, through which users can quickly check MLC status in NC system. If more advanced controlling is required, please refer to NC Series MLC Application Manual.

14

14.1 MLC Special M and D Command

The MLC (Motion Logic Control) and NC systems are two independent systems. MLC system performs knob and button controls, mechanical operations, and other electric logic controls, while NC system manages system and servo axis related functions. The MLC special M and D serve as the I/O interface between these two systems for data exchange and signal transmission. Output mentioned in this chapter refers to the signals sent to the NC system by MLC special M and D. Input refers to the signals sent to MLC special M and D by the NC system. The M letter prefixed commands are in bit format referring to signal 0 (OFF) or 1 (ON). The D prefixed ones are in word format referring to numerical values like 1000. MLC special M and D codes are all expressed in the form of M- and D- suffixed with four digits. Data exchanges between the two systems are divided into four groups:

1: MLC bit output from MLC to NC (special M, bit output)
2: MLC bit input from NC to MLC (special M, bit input)
3: MLC word output from MLC to NC (special D, word output)
4: MLC word input from NC to MLC (special D, word input)

14.2 Special M description for MLC bit output

14.2.1 Special M output for MLC and NC systems

Variable #1801~#1832 can be used in a machining program to read the signal status of MLC HMI output points. Variable #1801 ~ #1832 are paired with MLC Interface output points M1024 ~ M1055 respectively. For example, #1801 is paired with M1024, and so forth, for total 32 pairs. If M1024 output is On, the variable in the NC program #1801 will be 1, and this value will be 0 if M1024 output is Off.

Global Bit (MLC > NC)

Function name	Special M code	Variable ID	Function name	Special M code	Variable ID
HMI output 1	M1024	#1801	HMI output 17	M1040	#1817
HMI output 2	M1025	#1802	HMI output 18	M1041	#1818
HMI output 3	M1026	#1803	HMI output 19	M1042	#1819
HMI output 4	M1027	#1804	HMI output 20	M1043	#1820
HMI output 5	M1028	#1805	HMI output 21	M1044	#1821
HMI output 6	M1029	#1806	HMI output 22	M1045	#1822
HMI output 7	M1030	#1807	HMI output 23	M1046	#1823
HMI output 8	M1031	#1808	HMI output 24	M1047	#1824
HMI output 9	M1032	#1809	HMI output 25	M1048	#1825
HMI output 10	M1033	#1810	HMI output 26	M1049	#1826
HMI output 11	M1034	#1811	HMI output 27	M1050	#1827
HMI output 12	M1035	#1812	HMI output 28	M1051	#1828

Function name	Special M code	Variable ID	Function name	Special M code	Variable ID
HMI output 13	M1036	#1813	HMI output 29	M1052	#1829
HMI output 14	M1037	#1814	HMI output 30	M1053	#1830
HMI output 15	M1038	#1815	HMI output 31	M1054	#1831
HMI output 16	M1039	#1816	HMI output 32	M1055	#1832

14.2.2 Special M output for NC system

The special M in this section is for signal transmission from MLC to NC system. Using mechanical keys or knobs, the MLC program outputs special M signal to NC system to change NC modes or enable/disable functions. For example, users can enable M1060 output in MLC program to have NC system run the "Single block execution" function. Please refer to the table below for the special M used for NC system modes and functions.

Function name	Special M code	Description
System mode selection: 0 Auto execution (AUTO) 1 Edit (EDIT) 2 Manual input (MDI) 3 Hand wheel feeding (MPG) 4 Jog (JOG) 5 Fast feeding (RAPID) 6 Homing (HOME)	M1056 M1057 M1058 M1059	The NC system modes can be selected through M1056 ~ M 1059, which is represented by Bit 0 ~ Bit 3 in binary format. The binary number can be converted to decimals 0 ~ 6 referring to each system mode. For example, MPG mode is represented by decimal 3 (= binary number 0011) and its corresponding four bits in MLC are M1056 ~ M1059. Thus, the bit status of MPG mode is shown as below: M1056 ~ M1059 Binary table 0011 M1056 = ON M1057 = ON M1058 = OFF M1059 = OFF
Single block execution	M1060	In auto mode, program stops after one block is executed.
Cycle Start	M1061	Issue the auto execution signal
NC STOP	M1062	NC controller pauses after M1062 is triggered.
System STOP	M1063	The system stops operating.
Dummy execution	M1065	After M1065 is triggered, the movement speed F of G01 in auto mode will be set as the feed rate in register D1062.
Optional stop (M01 Pause)	M1066	Optional stop key. The controller pauses when M01 is executed in the program.
Single block skip ('/')	M1067	The program will skip the block with symbol '/' when this function is enabled.

Function name	Special M code	Description
Mechanical lock of multiple axes	M1068	Lock axis X, Y, and Z movement.
Lock axis Z	M1069	Lock axis Z movement.
Relieve axis limit	M1070	The limit signal of each axis will be ignored when this function is active.
Lock M, S, and T codes	M1071	Lock M, S, and T codes. The program will skip M, S, T codes in the execution.
Macro call initial preparation	M1074	The initial input of macro calling (only works in auto mode and with correct macro ID).
Macro call activation	M1075	Activate macro calling.
System reset	M1076	When M1076 is triggered, NC system will be reset. (MLC > NC)
MPG simulation	M1080	When executing the program, MPG can be used to control the speed of movement trails.
MST Code executed flag	M1152	When M1152 is triggered, NC system will be informed that M, S or T codes have completed their execution.
Tool magazine 1 moves forward	M1168	Tool magazine 1 moves forward. When M1168 is triggered, the standby tool pot (D1373) adds 1 to its value.
Tool magazine 1 moves backward	M1169	Tool magazine 1 moves backward. When M1169 is triggered, the standby tool pot (D1373) subtracts 1 from its value.
Tool 1 exchange	M1170	Exchange tool No. in tool magazine 1. Carry out tool No. exchange between the tool No. in use (D1374) and the standby tool No. (D1371).
Tool magazine 2 moves forward	M1172	Tool magazine 2 moves forward. When M1172 is triggered, the standby tool pot (D1377) adds 1 to its value.
Tool magazine 2 moves backward	M1173	Tool magazine 2 moves backward. When M1173 is triggered, the standby tool pot (D1377) subtracts 1 from its value.
Tool 2 exchange	M1174	Exchange tool data in tool magazine 2. Carry out tool No. exchange between the current spindle No. (D1378) and the tool No. set by G code (D1375).
Panel MPG pulse +	M1118	This is the trigger signal for forward movement when using the keys on the secondary control panel for MPG function. See D1040 for the enabling method of MPG function.

Function name	Special M code	Description
Panel MPG pulse +	M1119	This is the trigger signal for backward movement when using the keys on the secondary control panel for MPG function. See D1040 for the enabling method of MPG function.
Lock user 1 permission	M2934	M2934 can be used to lock the permission of user 1. This function only works when P10015 parameter (methods of granting permission) is set to 1.
Lock program editing	M2935	Prevent the program in controllers from being edited.

14

14.2.3 Special M output for NC axes

When the special M signals in this section are triggered, NC system will be instructed to conduct axis actions. For example, the forward jog of axis X will be enabled when M1216 is set to On. The table below lists the special M codes for the action controls of each NC axis:

Function name	Special M code	Function name	Special M code
Trigger flag of synchronous control	M1088	Axis X homing control	M1236
The slave axis X follows the master axis	M1089	Axis Y homing control	M1237
The slave axis Y follows the master axis	M1090	Axis Z homing control	M1238
The slave axis Z follows the master axis	M1091	Axis A homing control	M1239
The slave axis A follows the master axis	M1092	Axis B homing control	M1240
The slave axis B follows the master axis	M1093	Axis C homing control	M1241
The slave axis C follows the master axis	M1094	Relieve the 1st software limit of Axis X	M1248
Trigger flag of transfer command controls	M1098	Relieve the 1st software limit of Axis Y	M1249
Axis X receives command from master axis	M1099	Relieve the 1st software limit of Axis Z	M1250
Axis Y receives command from master axis	M1100	Relieve the 1st software limit of Axis A	M1251
Axis Z receives command from master axis	M1101	Relieve the 1st software limit of Axis B	M1252
Axis A receives command from master axis	M1102	Relieve the 1st software limit of Axis C	M1253
Axis B receives command from master axis	M1103	Lock axis X	M1257
Axis C receives command from master axis	M1104	Lock axis Y	M1258
Axis X forward jog control	M1216	Lock axis Z	M1259
Axis Y forward jog control	M1217	Lock axis A	M1260
Axis Z forward jog control	M1218	Lock axis B	M1261
Axis A forward jog control	M1219	Lock axis C	M1262
Axis B forward jog control	M1220	Axis X Servo Off	M1266
Axis C forward jog control	M1221	Axis Y Servo Off	M1267
Axis X backward jog control	M1226	Axis Z Servo Off	M1268
Axis Y backward jog control	M1227	Axis A Servo Off	M1269
Axis Z backward jog control	M1228	Axis B Servo Off	M1270
Axis A backward jog control	M1229	Axis C Servo Off	M1271
Axis B backward jog control	M1230	-	-
Axis C backward jog control	M1231	-	-

14

14.2.4 Special M output for spindle and MLC axes

Please refer to the following special M list for the controlling of spindle actions.

Function name	Special M code	Function name	Special M code
Spindle moves forward	M1120	Spindle positioning control	M1124
Spindle moves backward	M1121	Spindle returns from tapping	M1125
Spindle gear selection	M1122	-	-
	M1123	-	-

Note:
The selection of spindle gear ratio is presented by the combination of M1122 (Bit 0) and M1123 (Bit 1), and the bit range is 0 ~ 3 representing the four gear ratio (parameter P422 ~ P429). For example: 3 (= binary number 11) has to be selected for “gear ratio with numerator as 4 (parameter P428) and denominator as 4 (parameter P429)”. The corresponding two Bits in MLC are: M1122 = ON; M1123 = ON.

14.3 Special M Description for MLC bit input

14.3.1 Special M input for MLC and NC systems

Variable #1864~#1895 can be used in a NC program to read the signal status of MLC “HMI input points”. Variable #1864 ~ #1895 are paired with MLC interface input points M2080 ~ M2111 respectively. For example, #1864 is paired with M2080, and so forth, for total 32 pairs. If the variable #1864 = 1, M2028 in MLC is On, and if #1864 = 0, M2028 is Off.

Function name	Special M code	Variable ID	Function name	Special M code	Variable ID
HMI input 1	M2080	#1864	HMI input17	M2096	#1880
HMI input 2	M2081	#1865	HMI input 18	M2097	#1881
HMI input 3	M2082	#1866	HMI input 19	M2098	#1882
HMI input 4	M2083	#1867	HMI input 20	M2099	#1883
HMI input 5	M2084	#1868	HMI input 21	M2100	#1884
HMI input 6	M2085	#1869	HMI input 22	M2101	#1885
HMI input 7	M2086	#1870	HMI input 23	M2102	#1886
HMI input 8	M2087	#1871	HMI input 24	M2103	#1887
HMI input 9	M2088	#1872	HMI input 25	M2104	#1888
HMI input 10	M2089	#1873	HMI input 26	M2105	#1889
HMI input11	M2090	#1874	HMI input 27	M2106	#1890
HMI input 12	M2091	#1875	HMI input 28	M2107	#1891
HMI input 13	M2092	#1876	HMI input 29	M2108	#1892
HMI input 14	M2093	#1877	HMI input 30	M2109	#1893
HMI input 15	M2094	#1878	HMI input 31	M2110	#1894
HMI input 16	M2095	#1879	HMI input 32	M2111	#1895

14

14.3.2 Special M input for NC system

You can acquire the NC system's current status via the signals sent from NC system to MLC special M. And this signal can be used for status synchronizing between NC and MLC. The table below lists NC system status and the corresponding special M.

Function name	Special M code	Description
Machine started and system is ready	M2112	NC system is ready.
System alarm message	M2113	Alarm occurs in the NC system.
System emergency stop	M2114	System stops immediately after the EMG key is pressed.
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)
Macro call initialization completed	M2224	Macro call initialization completed (only works in auto mode and with correct macro ID)
Activating flag of macro call	M2225	Flag M2225 activates the execution of macro call.
Error flag of macro call	M2226	Flag M2226 indicates that error occurred in the macro calling.
Synchronous function in execution	M2227	NC system sends this signal when the synchronous function is in execution.
Transfer function in execution	M2228	NC system sends this signal when the transfer function is in execution.
Channel alarm message	M2240	Irregularity occurs in NC channel.
Auto execution (AUTO)	M2241	NC system sends this signal in AUTO mode.
Edit (EDIT)	M2242	NC system sends this signal in EDIT mode.
Manual input(MDI)	M2243	NC system sends this signal in MDI mode.
Hand wheel feed (MPG)	M2244	NC system sends this signal in MPG mode.
Jog (JOG)	M2245	NC system sends this signal in JOG mode.
Fast feed (RAPID)	M2246	NC system sends this signal in RAPID mode.
Homing (HOME)	M2247	NC system sends this signal in HOME mode.
Single block execution	M2249	NC system sends this signal when the program stops after executing single block.
Cycle Start	M2250	NC system sends this signal when the program starts running.
Pause	M2251	NC system sends this signal when the system paused.
M00 program stops	M2252	NC system sends this signal when M00 is executed.
M01 optional pause	M2253	NC system sends this signal when M01 is executed.

Function name	Special M code	Description
M02 program ends	M2254	NC system sends this signal when M02 is executed.
M30 program ends and returns	M2255	NC system sends this signal when M30 is executed.
Program ends	M2271	NC system sends this signal when the machining program ends.

14.3.3 Special M input for M, S, T codes

When M, S, and T codes are executed in a program, NC system will send the corresponding special M to MLC. For example, when M03 is executed in a program, M2208 in MLC will be set to On accordingly. Followings are the special M corresponding to M, S, T codes.

Function name	Special M code	Description
M Code Execution flag	M2208	When M codes are executed in the program, NC system will send the corresponding special M to MLC and M2208 will be set to On. The specified device will be enabled to carry out its command. M2208 will be set to Off when MST Code completed flag (M1152) in MLC is triggered. NC system will pause while waiting for the flag M1152, and the execution will be resumed after the flag M1152 is triggered. The M codes mentioned here do not include M00, M01, M02, M30, M98, M99 and the M code specified as macro.
S Code Execution flag	M2209	When S codes are executed in the program, NC system will send the corresponding special M to MLC and M2209 will be set to On. The specified device will be enabled to carry out its command. M2209 will be set to Off when MST Code completed flag (M1152) in MLC is triggered. NC system will pause while waiting for the flag M1152, and the execution will be resumed after the flag M1152 is triggered. This function will not work when using the S code specified as macro.
T Code Execution flag	M2210	When T codes are executed in the program, NC system will send the corresponding special M to MLC and M2210 will be set to On. The specified device will be enabled to carry out its command. M2210 will be set to Off when MST Code completed flag (M1152) in MLC is triggered. NC system will pause while waiting for the flag M1152, and the execution will be resumed after the flag M1152 is triggered. This function will not work when the T code has been used for macro calling. Flag M2210 is related to the station ID in the tool magazine. The flag can be triggered only when the value of T code is within the range of station ID specified in the tool magazine.

14

14.3.4 Special M input for NC axes

The corresponding special M code will be triggered when the hardware signal is input. The hardware limit signals are transmitted via the port “AXIS 1 ~ 4” at the back of the NC controllers. When the homing process of the axis has completed, the special M indicating “Homing completed” will be set to On. The table below lists the special M codes corresponding to the hardware limit of each axis and its homing signal and axis movement.

Function name	Special M code	Function name	Special M code
Port 1 positive hardware limit	M2144	Axis X homing completed	M2272
Port 1 axis negative hardware limit	M2145	Axis Y homing completed	M2273
Port 1 axis home signal	M2146	Axis Z homing completed	M2274
Port 2 axis positive hardware limit	M2148	Axis A homing completed	M2275
Port 2 axis negative hardware limit	M2149	Axis B homing completed	M2276
Port 2 axis home signal	M2150	Axis C homing completed	M2277
Port 3 axis positive hardware limit	M2152	Axis X positioned at the second reference point	M2286
Port 3 axis negative hardware limit	M2153	Axis Y positioned at the second reference point	M2287
Port 3 axis home signal	M2154	Axis Z positioned at the second reference point	M2288
Port 4 axis positive hardware limit	M2156	Axis A positioned at the second reference point	M2289
Port 4 axis negative hardware limit	M2157	Axis B positioned at the second reference point	M2290
Port 4 axis home signal	M2158	Axis C positioned at the second reference point	M2291
Port 5 positive hardware limit	M2160	Axis X is moving	M2320

Function name	Special M code	Function name	Special M code
Port 5 axis negative hardware limit	M2161	Axis Y is moving	M2321
Port 5 axis home signal	M2162	Axis Z is moving	M2322
Port 6 positive hardware limit	M2164	Axis A is moving	M2323
Port 6 axis negative hardware limit	M2165	Axis B is moving	M2324
Port 6 axis home signal	M2166	Axis C is moving	M2325

14.3.5 Special M output for spindle, tool magazine and MLC axes

In rigid tapping or before tool exchanges, the following special M can be used to check the status of spindle positioning and speed.

Function name	Special M code	Function name	Special M code
Spindle reaches the target speed	M2256	Spindle is in rigid tapping mode	M2259
Spindle reaches zero speed	M2257	Rigid tapping interruption	M2260
Spindle positioning completed	M2258	Spindle homing completed	M2281

14.4 Special D Description for MLC register output

14.4.1 Special D output for MLC and NC systems

Variable #1833~#1848 can be used in a machining program to read the register values of MLC “HMI output registers”. Variable #1833 ~ #1848 are paired with MLC Interface output points D1024 ~ D1039 respectively. There are total 16 pairs. For example, #1833 in NC program pairs with MLC’s output register D1024 in MLC. If the output value of D1024 is 100, the value of #1833 will be 100 accordingly. That is, the value of #1833 varies with the value of register D1024. Please refer to the table below for MLC output registers and the corresponding variables in NC system (MLC > NC):

Function name	Special D code	Variable ID	Function name	Special D code	Variable ID
HMI output register 1	D1024	#1833	HMI output register 9	D1032	#1841
HMI output register 2	D1025	#1834	HMI output register 10	D1033	#1842
HMI output register 3	D1026	#1835	HMI output register 11	D1034	#1843
HMI output register 4	D1027	#1836	HMI output register 12	D1035	#1844
HMI output register 5	D1028	#1837	HMI output register 13	D1036	#1845

14

Function name	Special D code	Variable ID	Function name	Special D code	Variable ID
HMI output register 6	D1029	#1838	HMI output register 14	D1037	#1846
HMI output register 7	D1030	#1839	HMI output register 15	D1038	#1847
HMI output register 8	D1031	#1840	HMI output register 16	D1039	#1848

14.4.2 Special D output for NC system

The special D in this section is for data transmission from MLC to NC system. It is to set MPG function and feed rate. Please refer to the chart below for more details.

Function name	Special D code	Description
Number of the processed products	D1022	It can be set in the Process screen or by MLC input.
Number of the processing target	D1023	It can be set in the Process screen or by MLC input.
MPG operation mode ID	D1040	This function is to set the MPG operation mode. When D1040 is set to 0, it is for external MPG. If it is set to 10, the MPG is controlled by the secondary control panel. The trigger flags of the pulse control are M1118 and M1119.
MPG operation channel selection	D1041	D1041 helps to designate the MPG operation channel. The default value is 0.
Set MPG pulse magnification	D1042	D1042 is to set MPG pulse magnification, ×1, ×10, and ×100. And it usually works with the actual MPG. When rotate one MPG scale, the actual movement is 0.001 mm (the min. unit) multiplies the pulse magnification. For example, if the magnification is 1, the actual moving amount is 1×0.001 = 0.001 mm/scale.
MPG axis selection	D1043	You can select the axis to be moving via MPG. It is set that 0 = axis X, 1 = axis Y, and 2 = axis Z.
Calling macro file name	D1111	Specify the call macro file name as O9xxx. For example, if D1111 writes K9100, the system will call macro named O9100.

14.4.3 Special D output for NC axes

The special D in this section is for data transmission from MLC to NC system. It is to set the speed rate for various NC functions. Please see the table below for the relevant special D.

Function name	Special D code	Description
Adjustment of cutting feed rate	D1056	Set the adjustment ratio of the cutting feed rate (F) in NC programs. If F is set to 1000 and the current value of D1056 is 50, it means the actual command speed is F500 mm/min (1000 x 50%).
Speed adjustment of rapid movement	D1058	Set the adjustment ratio of G00's max. speed (rapid movement). For example, if the speed of rapid movement is 6000, and D1058 is set to 50, it means the actual speed of G00 will be 3000 mm/min (= 6000 x 50%).
Spindle speed adjustment rate	D1060	Set the adjustment ratio of the S value specified in the program. For example, if S1000 is given in the program and D1060 is set to 30, it means the actual spindle speed is S300 r/min.
Set the speed of Jog and Dry run	D1062	Set movement speed F for dry run in JOG or AUTO mode. For example, set special D to 50 indicates F50 (mm/min) with a range of 0 ~ 65535 mm/min.

14.5 Special D Description for MLC register input

14.5.1 Special D input for MLC and NC systems

Variable #1896~#1911 can be used in a machining program to access the signal value of MLC "HMI input points". Variable #1896 ~ #1911 are paired with MLC Interface input points D1336~D1351 respectively. There are total 16 pairs. For example, #1896 is paired with D1336: If #1896 = 101 in NC program, the value of D1336 in MLC is 101 as well. That is, D1336 in MLC varies with variable #1896 in NC system. Please refer to the table below for MLC input registers and the corresponding variables in NC system (NC > MLC):

Function name	Special D code	Variable ID	Function name	Special D code	Variable ID
HMI input register 1	D1336	#1896	HMI input register 9	D1344	#1904
HMI input register 2	D1337	#1897	HMI input register 10	D1345	#1905
HMI input register 3	D1338	#1898	HMI input register 11	D1346	#1906
HMI input register 4	D1339	#1899	HMI input register 12	D1347	#1907
HMI input register 5	D1340	#1900	HMI input register 13	D1348	#1908

14

Function name	Special D code	Variable ID	Function name	Special D code	Variable ID
HMI input register 6	D1341	#1901	HMI input register 14	D1349	#1909
HMI input register 7	D1342	#1902	HMI input register 15	D1350	#1910
HMI input register 8	D1343	#1903	HMI input register 16	D1351	#1911

14.5.2 Special D input for M, S, T codes

The special D in this section, corresponding to the variables in NC system, is sent to MLC by NC program. For example, when the NC program is executing M03, D1368 in MLC will be 3. The special D can be used to access information such as the tool No. in execution, spindle speed, and cutting feed rate. Please see the table below for more details.

Function name	Special D code	Description
M code data	D1368	When M code is executed in a program, the value of M code will be saved in register D1368. For example, when executing M3 command, the value of D1368 is 3. The M codes mentioned here do not include M00, M01, M02, M30, M98, M99 and the M code used for macro call..
S code data	D1369	When the program encounters S code, the S code value will be saved in register D1369.
T code data (command)	D1370	When T code is executed in a program, the T code value will be saved in register D1370. It will not work when the T code is used for macro calling. The T code specified by the program will correctly display in D1370 only if it is within the range of station ID specified in the tool magazine.
Standby tool No. (tool magazine 1)	D1371	The Register Magazine in tool magazine 1 displays the tool No. corresponding to the standby tool pot (D1373).
Tool pot offset (tool magazine 1)	D1372	It is used to save the tool pot offset between the positions specified in D1370 (T code data) and D1371 (standby tool No.) in tool magazine 1. When the tool magazine is moving forward and backward during tool exchange (M1172/1173), 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 No. in tool magazine 1.
Tool No. in use (tool magazine 1)	D1374	The tool No. that is currently in use in tool magazine 1.
Standby tool No. (tool magazine 2)	D1375	The Register Magazine in tool magazine 2 displays the tool No. corresponding to the standby tool pot (D1377).

Function name	Special D code	Description
Tool pot offset (tool magazine 2)	D1376	It is used to save the offset between the positions specified in D1370 (T code data) and D1375 (standby tool No.) in tool magazine 2. When the tool magazine is moving forward and backward during tool exchange (M1172/1173), 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 No. in tool magazine 2.
Tool No. in use (tool magazine 2)	D1378	The tool No. that is currently in use in tool magazine 2.
Feed rate	D1379	Access the feed rate during cutting.
Spindle speed	D1380	Access spindle speed.

14.5.3 Special D input for NC axes

These special D signals are transmitted from NC to MLC, which are used for accessing the mechanical coordinates.

Function name	Special D code	Description
Axis X mechanical coordinates	D1384	The current mechanical coordinates of axis X.
Axis Y mechanical coordinates	D1386	The current mechanical coordinates of axis Y.
Axis Z mechanical coordinates	D1388	The current mechanical coordinates of axis Z.

(This page is intentionally left blank.)

Troubleshooting 15

This chapter includes description about the alarms and troubleshooting methods for NC system. Users can search this chapter for the methods of handling NC system related malfunctions.

15

15.1 Overview of NC controller alarms

15.1.1 PLC system alarms (Code：0x1200 ~ 0x1300)

Alarm code	Name	Causes and troubleshooting
0x1200	Accessing NC memory in error	1. An error has occurred when MLC accesses NC memory. 2. Please restart the controller or send it back for servicing.
0x1201	System is not ready	1. The startup procedure of NC system has not completed. 2. Please restart the controller or send it back for servicing.
0x1202	Buffer memory error	1. NC Buffer memory error or data is not ready. 2. Please restart the controller or send it back for servicing.
0x1203	Output port not exist	1. NC output port does not exist. 2. Please make sure the setting of axial parameter is correct.
0x1204	MLC code clearance error	1. Fail to clear the code of MLC program. 2. Please send the controller back for servicing.
0x1205	MLC flash memory error	1. An error has occurred when writing in MLC code to the flash memory. 2. Please restart the controller or send it back for servicing.
0x1206	SRAM error	1. An error has occurred when writing in SRAM. 2. Please send the controller back for servicing.
0x1207	Host I/O channel error	1. An error has occurred when accessing host I/O. 2. Please restart the controller or send it back for servicing.
0x1208	Remote I/O channel error	1. An error has occurred when accessing remote I/O. 2. Please restart the controller or send it back for servicing.
0x120A	NC PAR error	1. NC parameter is not set up or initialized. 2. Please re-initialize the parameter.
0x120B	Compen. PAR error	1. Compensation parameter memory write-in error. 2. Please reload the compensation parameters.
0x120C	Compen. PAR clearance error	1. An error has occurred when clearing the compensation parameter memory. 2. Please reload the compensation parameters.

Alarm code	Name	Causes and troubleshooting
0x120D	Compen. PAR write-in error	1. An error has occurred when writing in compensation parameter memory. 2. Please reload the compensation parameters.
0x120E	PAR initialization error	1. An error has occurred when initializing parameters. 2. Please re-initialize the parameter.
0x120F	Memory clearance error	1. Unable to clear the memory normally. 2. Please restart the controller or send it back for servicing.
0x1210	Memory write-in error	1. An error has occurred when writing in or initializing the memory. 2. Please restart the controller or send it back for servicing.
0x1211	Servo is not found	1. Wrong parameter setting. 2. Please verify the parameter settings.
0x1212	Servo PAR type error	1. Wrong parameter setting. 2. Please verify the parameter settings.
0x1213	DMCNET initialization error	1. DMCNET initialization error. 2. Please ensure the connection of DMCNET is securely connected.
0x1214	Non-volatile memory error	1. An error has occurred when writing in and reading the non-volatile memory. 2. Please restart the controller or send it back for servicing.
0x1216	MLC PRG error	1. Check the MLC program; 2. Please reload the MLC program.
0x1217	MLC PAR setting error	Modify MLC parameter setting.
0x1300	Network com. error	1. Please check the network connection. 2. Please restart the controller or send it back for servicing.
0x1E00	Servo error	1. A servo alarm has occurred. 2. Please check the servo status or replace a new servo drive.
0x1F00	Remote I/O error	1. Remote I/O error. 2. Please make sure the connection for remote I/O is well or replace a new remote I/O board.

15.1.2 NC alarms (Code: 0x4200 ~ 0x4300)

Alarm code	Name	Causes and troubleshooting
0x4200	Homing is required	1. Homing is not executed yet, and please execute homing procedure; 2. Please check the wiring and parameter settings.
0x4201	Absolute origin setup.	1. Set the origin position; 2. Please check the battery power.
0x4300	MLC accessing NC data in error	1. MLC is not ready or an error occurs when accessing the memory. 2. Please restart the controller or send it back for servicing.
0x4301	MLC is not ready	1. MLC is not ready. 2. Please restart the controller or send it back for servicing.
0x4302	I/O module PRG clearance error	1. I/O module PRG clearance error. 2. Please re-install the program.
0x4303	I/O module PRG write-in error	1. I/O module PRG write-in error. 2. Please re-install the program.
0x4304	NC system PRG clearance failed	1. Fail to clear the NC system program. 2. Please re-install the program.
0x4305	NC system PRG installation failed	1. Fail to install the NC system program. 2. Please re-install the program.
0x4306	Macro clearance failure	1. Fail to clear the macro program. 2. Please re-install the program.
0x4307	Macro installation failure	1. Fail to install the macro program. 2. Please re-install the program.
0x4308	G code loading error	1. An error has occurred when loading G code. 2. Please verify the machining program.
0x4310	I/O module PRG uninitialized	1. I/O module program is not initialized. 2. Re-install I/O module program.
0x4311	I/O module initialized data length error	1. I/O module memory error. 2. Please restart the controller or send it back for servicing.
0x4312	I/O module data content error	1. I/O module memory error. 2. Please restart the controller or send it back for servicing.
0x4313	I/O module status error	1. I/O module status error. 2. Please ensure the I/O board is firmly installed.

Alarm code	Name	Causes and troubleshooting
0x4314	I/O module PRG config. error	1. Program configuration of I/O module is in error. 2. Please ensure the I/O board is firmly installed.
0x4315	I/O module serial interface error	1. Reading and write-in of I/O module interface is in error. 2. Please ensure the I/O board is firmly installed.
0x4316	I/O module interface access error	1. An error has occurred when reading the I/O board interface. 2. Please send the controller back for servicing.
0x4317	NC system cmd error	1. An error has occurred when reading and writing NC system command. 2. Please send the controller back for servicing.
0x4318	NC PAR loading failure	1. NC parameter error or MLC is not ready. 2. Please restart the controller or send it back for servicing.
0x4319	NC PAR does not exist	1. NC parameter error or MLC is not ready. 2. Please restart the controller or send it back for servicing.
0x431A	Tool magazine axis error	1. Tool magazine axis is not defined or defined repeatedly. 2. Please verify the parameter settings.
0x431B	NC PAR error	1. NC parameter error or MLC is not ready. 2. Please restart the controller or send it back for servicing.

15

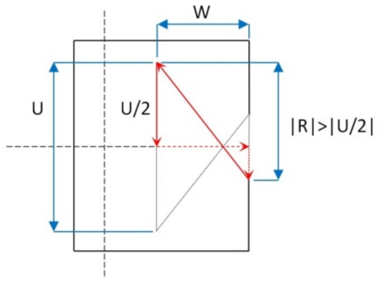
15.1.3 Channel alarms (Code: 0xA000 ~ 0xD000)

Description of illegal G code line number and error messages:

Alarm code	Name	Causes and troubleshooting
0xB000	Illegal G code line number	1. The G code line number is illegal. 2. Please verify the machining program.
0xB001	Illegal G code length	1. The G code length is illegal. 2. Please verify the machining program.
0xB002	G code file is not found	1. G code file is not found. 2. Please verify file contents.
0xB003	Invalid name of the loaded file	1. The name of the loaded file is invalid. 2. Please reload the program file.
0xB005	Workpiece coord. clearance error	1. Computing error of workpeice coordinates. 2. Please reset workpiece coordinates.
0xB006	Workpiece coord. computing error	1. Reading workpiece coordinates is in error 2. Please reset workpiece coordinates.
0xB007	Conflicting servo port settings	1. Station number for servo port repeats. 2. Please verify the parameter settings.
0xB008	Memory overlapping	1. Conflict has occurred while transferring commands. 2. Please restart the system.
0xB009	G code buffer zone error	1. G code buffer zone is in error. 2. Please reload the machining program.
0xB00A	Invalid interpolator cmd index	Please press the reset key, and load the program file again.
0xB00B	Interpolator cmd buffer zone access error	Please press the reset key, and load the program file again.
0xB00C	Feed rate is not defined	1. G code error. 2. Please check G code and revise the program.
0xB00D	Invalid arc radius	1. G code error. 2. Please check G code and revise the program.
0xB00E	Invalid tool ID selection	1. G code error. 2. Please check G code and revise the program.
0xB00F	Servo No. differs from PAR setting	Check if the parameter setting and the actual number of the connected axis is the same.
0xB010	Breakpoint number not found	Check if G code contains the label or line being searched; revise the program.
0xB014	Incorrect tool compen. radius	1. G code error. 2. Please check G code and revise the program.
0xB015	Sync cmd error	Check the MLC program procedure.

Alarm code	Name	Causes and troubleshooting
0xB017	Cutter specifying error	Make sure the cutter number is within the specified range.
0xB018	Switching transition cmd prohibited	1. Please check G code and revise the program. 2. Check the MLC program procedure.
0xB019	Cmd error	Axis cannot be moved; please check the procedure.
0xB01A	Data full	Please check G code and revise the program.
0xB01B	Spindle does not operate	The spindle does not operate during machining.
0xB01C	Spindle speed error	1. Spindle speed is too fast. 2. Please revise G code program.
0xB020	Emergency stop	1. Emergency stop is pressed. 2. Please check the connection of emergency stop.
0xB021	Can't figure out chamfer/ round corner	Chamfer/ round corner cannot be figured out.
0xB100	Radius compen. path interference	Please verify the path of tool compensation.
0xB101	Cancel radius compen. in arc	1. G code error. 2. Please check G code and revise the program.
0xB102	Enable radius compen. in arc	1. G code error. 2. Please check G code and revise the program.
0xB103	Radius interference	1. G code error. 2. Please check G code and revise the program.
0xB104	Tool compen. amount too small	1. G code error. 2. Please check G code and revise the program.
0xB105	Left & right compen. switch error	1. G code error. 2. Please check G code and revise the program.
0xB106	G31 is used when tool compen.	G31 jump function is not allowed during tool compensation, please revise the program.
0xB108	NURBS interpolation error	The file format is wrong or the first control position does not match.
0xB301	Thread cutting pitch error	1. If the commuting result of variable-pitch thread cutting is smaller than 0, this error occurs. 2. Please check G code and revise the program.
0xB302	Spindle speed too fast	The feedrate of thread cutting is too fast. Please reduce the spindle speed.
0xB600	Invalid G code ID	1. G code error. 2. Please check G code and revise the program.
0xB601	Too many subroutine nests	1. The subroutine calls too many programs. 2. Please revise the program to reduce the time of calling.

Alarm code	Name	Causes and troubleshooting
0xB602	No G code symbol	1. No G code symbol. 2. Please check G code and revise the program.
0xB603	Invalid variable symbol	1. Invalid variable symbol. 2. Please check G code and revise the program.
0xB604	Illegal G code symbol	1. Illegal G code symbol 2. Please check G code and revise the program.
0xB605	Workpiece coordinates data missing	1. The data of workpiece coordinates is missing. 2. Please press the reset key or restart the controller.
0xB606	Subroutine calling error	1. Subroutine does not exist. 2. Please revise the program.
0xB607	Subroutine file name err	1. Subroutine file name is invalid 2. Please revise the program.
0xB608	Subroutine nesting error	1. Number of the subroutine nest exceeds the range. 2. Please revise the program.
0xB60A	Syntax error of G04	1. The syntax for pause time is in error. 2. Please check G code and revise the program.
0xB60C	Arc magnification error	The syntax for arc magnification is in error
0xB60D	Middle point homing err	Intermediate point for homing is not defined
0xB60E	Homing when cycle mach.	Homing is not allowed during cycle machining; please revise the program.
0xB60F	G54 augmented code err	Please verify the augmented code and revise it based on the specified range.
0xB650	Undefined G10 function	The undefined function of G10 is used; please revise the program.
0xB651	G10 PAR range err	Invalid parameter range, please revise the program.
0xB652	No spindle speed for cycle	Spindle speed for the cycle is not specified; please revise the program.
0xB653	No feed rate for cycle	Feed rate for the cycle is not specified; please revise the program.
0xB654	Illegal cycle instruction	Cycle instruction is not defined; please revise the program.
0xB6A1	Finish turning path not found	The command cannot find the starting/ending number of the specified finish turning path; please revise the program.
0xB6A2	Finish turning path not specified	The finish turning path is not specified with starting/ending number; please revise the program.

Alarm code	Name	Causes and troubleshooting
0xB6A3	Taper error in single turning cycle cmd	Taper setting in single turning cycle command is incorrect; please revise the program. In turning cycle command, when the radius setting value (R) is greater than the moving distance (U/2), this alarm will occur. 
0xB6A4	Turning straight angel cmd cannot calculate	Turning straight angel command cannot calculate; please check and revise the program.
0xB6A5	Turning drilling and tapping cmd error	Turning drilling and tapping command is in error; please check and revise the program.

15

15.1.4 Error message of macro configuration

Alarm code	Name	Causes and troubleshooting
0xB610	Invalid macro VAR type	Please check the macro and revise the program.
0xB611	Macro cmd is not found	Please check the macro and revise the program.
0xB612	Invalid macro cmd line No.	N is not found when executing GO TO command. Please revise the program.
0xB613	Marco bit setup error	Please check the macro and revise the program.
0xB614	Divided by zero in macro	Please check the macro and revise the program.
0xB615	Macro cmd is too long	Macro command has exceeded the length, please revise the program.
0xB616	No macro cmd operand	Please check the macro and revise the program.
0xB617	Macro cmd error	Please check the macro and revise the program.
0xB618	Macro syntax error	Macro command not found, please revise the program.
0xB619	Macro operand syntax err	1. Macro operand syntax error. 2. Please check the macro and revise the program.
0xB61A	Illegal macro cmd	Please check the macro and revise the program.
0xB61B	GOTO tag is not found	Check the syntax of GOTO and revise the program.
0xB61C	No line No. given by GOTO	Please revise the program.
0xB620	User-defined macro alarms	The macro alarms defined by the users occur.
0xB621	Illegal look-ahead stop cmd	Please revise the program. The first block shall not contain a stop command for look-ahead or other commands.
0xB623	The feedrate set as negative value	1. Please check if the feedrate is set as negative value. 2. Please revise G code program.
0xB630	Excess following error	1. Check the servo connection. 2. Make sure the setting of servo parameter is correct.
0xB631	Hardware limit error	1. Please revise the program. 2. Check if the limit switch is valid or polarity setting is correct.
0xB632	The 1st software limit error	1. Please revise the program. 2. Check the parameter setting of the first software limit

Alarm code	Name	Causes and troubleshooting
0xB634	The 2nd software limit error	1. Please revise the program. 2. Check the parameter setting of the second software limit.
0xB636	Origin sensor error	1. Make sure the installation and the polarity settings of the origin switch are correct. 2. Ensure the parameter setting of the distance for looking the origin is correct.
0xB640	Temperature sensor err 1	1. Make sure the power supply of the module is normal. 2. Make sure each connector is properly connected.
0xB641	Temperature sensor err 2	1. Make sure the power supply of the module is normal. 2. Make sure each connector is properly connected.
0xB642	Temperature sensor err 3	1. Make sure the power supply of the module is normal. 2. Make sure each connector is properly connected.
0xB643	Temperature sensor err 4	1. Make sure the power supply of the module is normal. 2. Make sure each connector is properly connected.

15.1.5 HMI alarms (Code: 0x3010 ~ 0x3FFF)

Description of HMI interface error messages:

Alarm code	Name	Causes and troubleshooting
0x3010	HMI com. interface error	1. An error has occurred while creating HMI communication interface. 2. Please restart the controller or send it back for servicing.
0x3011	HMI com. memory zone error	1. An error has occurred while creating HMI com. memory zone. 2. Please restart the controller or send it back for servicing.
0x3012	HMI interface cmd zone error	1. An error has occurred while creating HMI interface cmd zone. 2. Please restart the controller or send it back for servicing.
0x3013	HMI interface memory error	1. HMI interface memory error. 2. Please restart the controller or send it back for servicing.
0x3014	HMI interface com. port error	1. HMI interface com. port error. 2. Please restart the controller or send it back for servicing.
0x3015	MLC interface memory error	1. MLC interface memory error. 2. Please restart the controller or send it back for servicing.
0x3016	HMI file transmission error	1. HMI file transmission error. 2. Please restart the controller or send it back for servicing.
0x3017	HMI data transmission error	1. HMI data transmission error. 2. Please restart the controller or send it back for servicing.
0x3100	Illegal file name	1. Check if the file exists; 2. Please rename the file.
0x3101	Subroutine nests called overrange	Reduce the subroutine nests being called.
0x3102	Non-G code character contained	1. Please check G code and revise the program. 2. Delete the symbol that is not allowed in G code.
0x3103	Memory diagnosis error	Please restart the controller or send it back for servicing.
0x3200	Internal PAR CRC error	Memory zone of system internal parameter is in error. Please use system recovery function or send the controller back for servicing.

Alarm code	Name	Causes and troubleshooting
0x3201	MLC PRG error	Memory zone of system MLC program is in error. Please import the MLC program again or send the controller back for servicing.
0x3202	CF card reading failure	No CF card inserted or invalid CF card is inserted.
0x3203	PAR backup failure	Ensure CF card is inserted properly and has adequate free space.
0x3204	MLC backup failure	Ensure CF card is inserted properly and has adequate free space.
0x3205	Expired. Machined locked	Machine expired, please contact the distributor to unlock or postpone the expiry date.
0x3206	PAR value exceeds the range	1. Please check all the parameter values of the system and make sure they are within the range. 2. Revise the parameter values that are not within the range.
0x3207	Function library loading error	1. An HMI interface cmd zone error occurred when loading the function library for connecting with the external device via RS-485. 2. Please make sure the setting of Base Port in ScreenEditor is correct and reload the program of software interface.
0x3208	Machine to be locked	1. Expiry date is close; the machine will be locked when expired and machining cannot be done. 2. Please contact the distributor for unlocking the machine or extension.
0x3209	Update complete; restart the system	Firmware update of the system is complete, please restart the controller.
0x320A	Low battery	The system battery is too low, please change the battery.
0x3210	RS-485 com. Breakdown	Ensure the connection via RS-485 between the controller and external device is well.
0x3211	Load COMM PORT DLL error	The software panel failed to load COMM PORT DLL; please update the software panel again and correctly load the external device link.
0x3212	Create COMM PORT error	An error has occurred when loading COMM PORT DLL; please update the software panel again and make sure the external link related setting is correct.

(This page is intentionally left blank.)

Revision History

Release date	Version	Chapter	Revision Contents
August, 2016	V1.0	N/A	First Edition
October, 2016	V2.0	Chapter 1	Add descriptions for keys that have two characters.
		Chapter 6 ~ Chapter 13	Use $\boxed{\text{Framed Text}}$ to indicate the keys in primary control panel. Use **boldface letter** to indicate the function key.

For other relevant information about NC series user manual for operation and maintenance, refer to the manual below:

(1) NC Series Command Guidelines (September, 2016)

(2) NC Series MLC Application Manual (September, 2016)

(This page is intentionally left blank.)