Delta CNC Solution NC200 Series User Manual

Industrial Automation Headquarters
Delta Electronics, Inc.
Taoyuan Technology Center
No.18, Xinglong Rd., Taoyuan City,
Taoyuan County 33068, Taiwan
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

Deltronics (The Netherlands) B.V.
Eindhoven Office
De Witbogt 20, 5652 AG Eindhoven, The Netherlands
TEL: 31-40-2592850 / FAX: 31-40-2592851

DELTA_IA-CNC_NC200_UM_EN_20150918

www.deltaww.com

Table of Contents

Chapter 5 Program (PRG) Group

Chapter 6 Offset (OFS) Group

Chapter 7   Diagnosis (DGN) Group

Chapter 8   Alarm (ALM) Group

Chapter 9   Graphic (GRA) Group

Chapter 10   Parameter (PAR) Group

Chapter 11   Table of G codes

Chapter 12   G codes description

Chapter 13   M commands

Chapter 14   MACRO & Variables

Chapter 15   Wiring

Chapter 16   Parameters

Chapter 17   Toubleshooting

(This page is intentionally left blank.)

Chapter 1 Table of Function Keys

1.1 Function keys of primary control panel

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$_\wedge$ … 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

Name	Description	Supported Mode
⬆; ⬇, ⬅& ➡#	Arrow keys (Up, Down, Left and Right) (computing symbol)	PRG, OFS, DGN group
HOME END	Jump to beginning (end) of word	PRG group
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 key	ALL

1.2   Function keys of secondary control panel

Name	Description
→]	Auto mode: For program execution
②]	Edit mode: For file management and program editing
ᴡᴡᴡ	Jog mode: For manually operate the machine tool
⊛	Hand wheel mode: Operate machine tools axis via hand wheel
▣]	Manual mode: For simple program input and execution
⊕	Homing mode: Promptly return to the original point
X← X→	X-axis forward, X-axis backward: In JOG mode, manually operate X-axis in forward or backward direction.
Y↖ Y↘	Y-axis forward, Y-axis backward: In JOG mode, manually operate Y-axis in forward or backward direction.
Z↑ Z↓	Z-axis forward, Z-axis backward: In JOG mode, manually operate Z-axis in forward or backward direction.
4↗ 4↙	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
+ − 切削/寸動	FEEDRATE/JOG%: Keys for increasing or decreasing the cut feeding speed and jog ratio
+ − 快速倍率	RAPID% (Rapid feeding ratio): Keys for increasing or decreasing the fast feeding ratio
+ − 主軸倍率	SP% (Spindle factor):Keys for increasing or decreasing the spindle speed ratio
→]	Single step pause: After enabling the function, the system stops execution when finish one single step.

Name	Description
	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 and execute M01 command to enable this function.
	Hand wheel simulation: During program execution, after this function is enabled, the hand wheel 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

Chapter 2 Mode Function

2.1   Auto mode (AUTO)

The system must be set to AUTO mode before a program is executed. In this mode, only executing program file is allowed; functions such as program editing and manually operating the axial movement are not available here.

2.2   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 cannot be executed and limiting axial operating direction is also not allowed.

2.3   Manual mode (MDI)

In manual mode, users can input a single block program in the screen of PRG group so as to execute this program. Generally, user can manually input the single block program to be executed instantly. Functions of program editing, program execution, or manually operating axis directions are not available in this mode.

2.4   Hand wheel mode (MPG)

In Hand wheel mode, it allows users to manually control the axis via external hand wheel. 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.

2.5   Jog mode (JOG)

Jog mode works with axial movement keys on the secondary control panel, which can control the speed and moving distance of each jog movement. With the rapid feeding key, the axial moving speed is set by the rapid factor. Both program execution and editing functions are unavailable in JOG mode.

2.6   Home mode (HOME)

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.

Chapter 3 Group Function Map

Position (POS) Function Map

POS / Position
- → [ABS.]
- → [REL.]
  - → [CLR ALL.]
  - → [CLR X.]
  - → [CLR Y.]
  - → [CLR Z.]
  - → [CLR A.]
  - → [CLR B.]
  - → [CLR C.]
- → [MECH]

Program (PRG)Function Map 1- [Edit mode]

PRG / Program
- → [NEW file]
- → [Copy file]
- → [Paste]
- → [DEL]
- → [SEL TOGL]
- → [CANCEL]
- → [FOLDER]
- → [Rename]
- → [FIND FILE]
- → [MERGE]
- → [Macro]

Program (PRG)Function Map 2- [Edit mode]

PRG → PRG / Program
- → [Copy]
- → [Paste]
- → [DEL]
- → [Undo]
- → [B start]
- → [B end]
- → [LABEL]
- → [STRING]
  - → [Next]
  - → [PREV.]

Program (PRG) Function map - [Auto mode]	Program (PRG) Function Map - [Manual mode]

PRG → [SF set]

Program

[START] → [RUN]

PRG → [Load]

Program

[Save]

[Clear]

Program (PRG) Function Map - [Jog mode], [Handwheel mode]

PRG → [SF set]

Program → [Teach Program] → [Fast Move]

[LINEAR]

[ARC] → [P1]

[P2]

[P3]

[DEL]

[SAVE]

[PLANE SEL]

[CREATE]

[MECH COOR./ABS COOR.]

Program (PRG) Function Map - [Homing mode]

PRG → [SF set]

Program

Offset (OFS) Function Map

```
OFS  →  [Coord.]  ──────────────→  [Auto]  →  [REL.CLR]
                                           →  [CLR All]
Offset                                     →  [SET L.]
                          [ABS.]  →        →  [SET L/2.]
                          [INC.]  →        ↳  [SET P.]

                          [SQUARE]  →  [X1]
                                    →  [X2]
                                    →  [Y1]
                                    →  [Y2]
                                    →  [Set]
                                    ↳  [Set Z]

                          [CIRCLE]  →  [P1]
                                    →  [P2]
            [CUTTER]  →  [ABS.]     →  [P3]
                      →  [INC.]     →  [Set]
                      →  [SET H.]   ↳  [Set Z]
                      ↳  [Clear]    →  [H/D]
                                    →  [Wear]
                                    →  [Life]
                                    ↳  [All]

            [MAGA]  →  [Maga 1]  →  [Set]
                                 →  [RST All]
                                 ↳  [LOCK]

                    ↳  [Maga 2]  →  [Set]
            [Macro]  →  [Local]  →  [RST All]
                     →  [Global] ↳  [LOCK]
                     →  [Hold]
                     ↳  [Expanded]
```

Diagnosis (DGN) Function Map 1

```
DGN ──→ [Machining] ──→ [Set]
Diagnosis           ──→ [CLR TIME]
                    ──→ [CLR NR.]

      ──→ [User      ──→ [Sys VAR.]
           VAR.]     ──→ [User      ──→ [US DEC.]
                          VAR.]     ──→ [DEC.]
                                    ──→ [S DEC.]
                                    ──→ [FLOAT]
                                    ──→ [DEL.]

      ──→ [MLC.] ──→ [Bit]  ──→ [X]
                            ──→ [Y]
                            ──→ [M]
                            ──→ [A]
                            ──→ [T]
                            ──→ [C]

                 ──→ [REG]  ──→ [T]
                            ──→ [C(16)]
                            ──→ [C(32)]
                            ──→ [D]
                            ──→ [V]
                            ──→ [Z]
                            ──→ [US DEC.]
                            ──→ [DEC.]
                            ──→ [S DEC.]
                            ──→ [FLOAT]

                 ──→ [DEV   ──→ [US DEC.]
                      MON.]  ──→ [DEC.]
                            ──→ [S DEC.]
                            ──→ [FLOAT]
```

To next page...   To next page...

Diagnosis (DGN) Function Map 2

...From last page   ...From last page

$\rightarrow$ **[EDITOR]** $\rightarrow$ **[LD.]**

$\rightarrow$ **[LDI]**

$\rightarrow$ **[LDP]**

$\rightarrow$ **[LDF]**

$\rightarrow$ **[OUT]**

$\rightarrow$ **[APP]**

$\rightarrow$ **[-]**

$\rightarrow$ **[|]**

$\rightarrow$ **[DEL V-LN]**

$\rightarrow$ **[ADD LN]**

$\rightarrow$ **[DEL LN]**

$\rightarrow$ **[DEL]**

$\rightarrow$ **[LABEL]**

$\rightarrow$ **[TABLE]**

$\rightarrow$ **[SYMBOL]** $\rightarrow$ **[X]**

$\rightarrow$ **[Y]**

$\rightarrow$ **[M]**

$\rightarrow$ **[A]**

$\rightarrow$ **[T]**

$\rightarrow$ **[C]**

$\rightarrow$ **[D]**

$\rightarrow$ **[P]**

$\rightarrow$ **[I]**

$\rightarrow$ **[DEL]**

$\rightarrow$ **[COPY]**

$\rightarrow$ **[PASTE]**

To next   To next   To next
page...   page...   page...

Diagnosis (DGN) Function Map 3

...From last page   ...From last page   ...From last page

```
                                              [SAVE]

                                              [IMPORT] ──→ [IMPORT]

                                              [EXPORT] ──→ [EXPORT]

                                              [JUMP TO]        [NEW
                                                               FILE]

                                              [SELECT]

                                              [CUT]

                                              [COPY]

                                              [PASTE]


                          [SET] ──────────→ [ON]

                          [JUMP TO]          [OFF]

                                             [RUN/STOP]


        [SYS          [VAR              [SYS VAR]
        MONI] ──────→ MONI] ─────────→

                      [I/O MONI]         [CH VAR]

                      [SRV              [AXIS VAR]
                      MONI]
                                         [IF VAR]

                                         [US DEC]


        [STATUS] ───→ [SYSTEM] ───────→ [BIT]

                      [FW SN] ─────────→ [HEX]

                      [HW SN] ─────────→ [S DEC]


        [PWD] ──────→ [S SCP] ─────────→ [UNLOCK]

                                         [LOCK]


                      [M SCP] ─────────→ [LOCK]

                                         [UNLOCK]

                                         [RST U1]

                                         [RST U2]
```

To next   To next
page...   page...

Diagnosis (DGN) Function Map 4

...From last page (left column)

...From last page (right column)

- [TUNING]
  - [U1 SCP]
    - [PWD CHG]
    - [LOCK/UNLOCK]
  - [U2 SCP]
    - [PWD CHG]
    - [LOCK/UNLOCK]
  - [EXPIRE]
    - [SETTING]
    - [RELEASE]
    - [EXP SCP]
      - [PWD CHG]
      - [LOCK/UNLOCK]
  - [NEX AX]
  - [READ]
  - [COMPUTE]
  - [WR GAIN]
  - [WR NOTH]
  - [RUN]
  - [JOG←]
  - [JOG→]
  - [POS 1]
  - [POS 2]
  - [TAP RIV]
  - [SERVO]
    - [TAP SET]

- [TEXT WR]
- [IMPORT]
  - [IMPORT]
- [EXPORT]
  - [EXPORT]
- [RECOVER]
  - [OK]
- [LOGO WR]

Alarm (ALM) Function Map

```
ALM ──┐     ┌──→ [Alarm]
      ├─────┤
Alarm ┘     └──→ [History] ──→ [CLR All]
```

Graphic (GRA) Function Map 1

```
GRA ──────→ [Cutting path] ──→ [X-Y/Y-Z/X-Z/X-Y-Z]
                            ──→ [Center]
Graphic                     ──→ [Zoom in]
                            ──→ [Zoom out]
                            ──→ [Draw]
                            ──→ [Stop Draw]
                            ──→ [Up]
                            ──→ [Down]
                            ──→ [Left]
                            ──→ [Right]

            [Cutting preview] ──→ [X-Y/Y-Z/X-Z/X-Y-Z]
                              ──→ [Center]
                              ──→ [Zoom in]
                              ──→ [Zoom out]
                              ──→ [Preview]
                              ──→ [Cancel preview]
                              ──→ [Up]
                              ──→ [Down]
                              ──→ [Left]
                              ──→ [Right]
```

Parameter (PAR) Function Map

PAR → [Process]

Parameter

→ [Operate]

→ [Maga]

→ [Spindle]

→ [Machine]

→ [Home]

→ [Network] → [Default]

→ [Comp] → [OK]

→ [um]

→ [um+]

→ [IMPORT]

→ [IMPORT+]

→ [System] → [Default]

→ [Color]

→ [MLC] → [Default]

→ [Color]

→ [Graphic] → [Default]

→ [Color]

→ [Servo] → [Read]

→ [Config] → [OK]

→ [Set RIO] → [OK]

→ [Search]

(This page is intentionally left blank.)

Chapter 4 Position (POS) Group

The POS group function is for displaying different coordinates, including data on mechanical, absolute, and relative coordinates.

Figure 4-1-1

4.1 Absolute Coordinates

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

(1) Press **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 **F1** key (ABS) to enter the absolute coordinates screen.

4.2   Relative coordinates

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

(1) Press **POS** key to enable the coordinate group function display. Items such as absolute, relative, and mechanical coordinates are shown on the function bar.

(2) Press **F2** key (REL) to enter the relative coordinates screen.

(3) Press **F1** (CLR ALL) in the lower layer function bar to clear relative coordinate values of all axes.

Press **F2** key (CLR X) to clear the relative coordinate value of the X-axis.

Press **F3** key (CLR Y) to clear the relative coordinate value of the Y-axis.

Press **F4** key (CLR Z) to clear the relative coordinate value of the Z-axis.

Press **F5** key (CLR A) to clear the relative coordinate value of A-axis.

Press **F6** key (CLR B) to clear the relative coordinate value of B-axis.

4.3   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 **POS** key to enable the coordinate group function display. Items such as absolute, relative, and mechanical coordinates are shown on the function bar.

(2) Press **F3** key (MECH) to enter the mechanical coordinates screen.

Chapter 5 Program (PRG) Group

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.
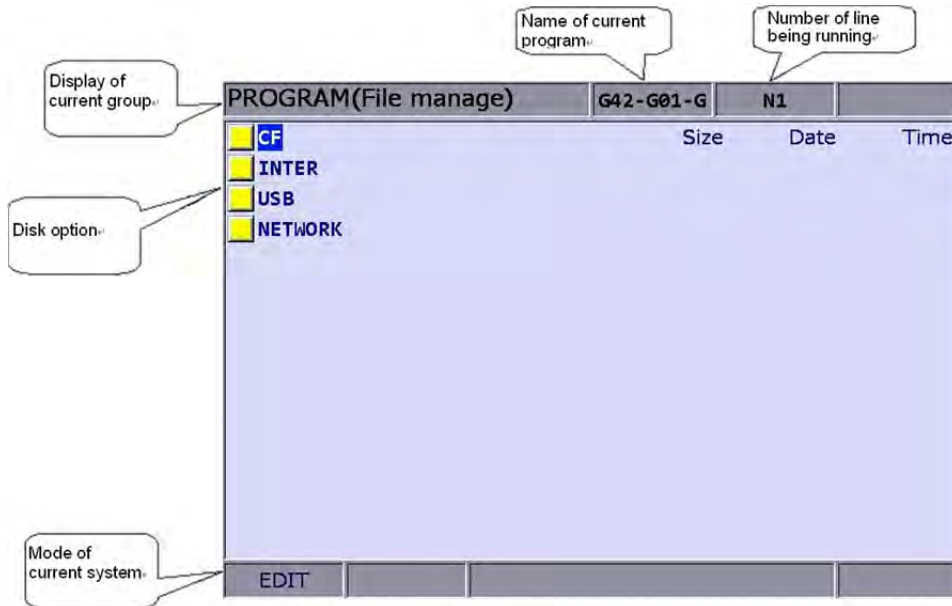
Figure 5-0-1

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

Press [↑]; [↓], or [PAGE UP] [PAGE DN] keys to move the cursor in the screen of [File manage] and press [ENTER] key to enter the second or third layer to select G code files.

Select the desired G code file and press [ENTER] key to enter the file editing screen.

Note: FAT 32 format for USB drive is recommended.

5.1   Network Setting

The NETWORK function remotely connects to a PC through Ethernet. With <u>CNC Network</u> <u>software</u>, this remote connection enables one PC to execute the function of online file management with multiple NC200 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 NC200 system and PC before using the networking function. See below for setup steps:

NC200 system communication protocol: Screen of PAR group → Screen of Ethernet.

Parameters for network setup		
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 computer 4	Length: ×××・×××・×××・××× 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 5-1-1) or **CNC Network software → Options**

Network setup in PC operating system:
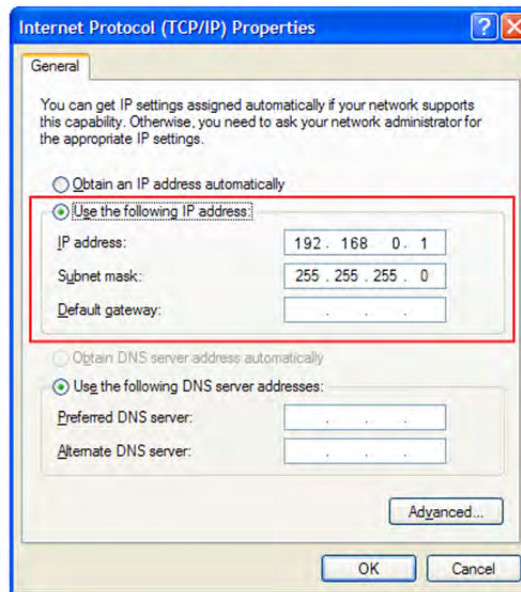
Figure 5-1-1

Steps: (a) Select **Use the following IP address** and then enter the address below:
IP address: 192 . 168 . 0 . 1
Subnet mask: 255 . 255 . 255 . 0
(b) Press **OK** to complete the setting.

Network setup for Network software:
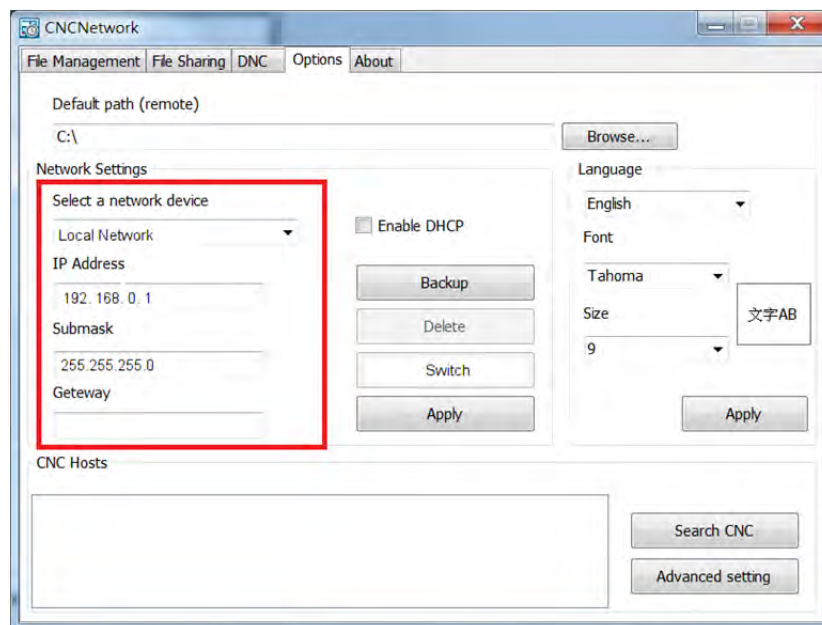
Figure 5-1-2

Steps: (a) 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

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

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 NC200 system.
(2) Start the **CNC Network software**.
(3) Click on the Function bar > **DNC**.
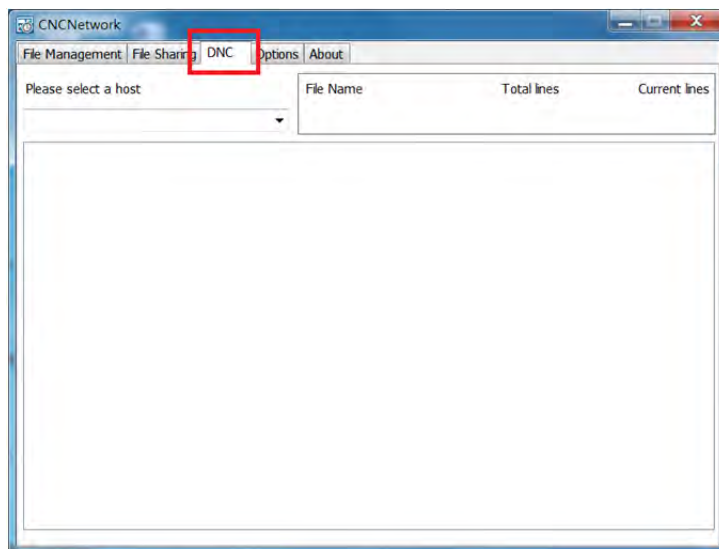
Figure 5-1-3

(4) Enter the **Edit mode** of NC200 system and then enter the top layer NETWORK\Option in [File manage].

Figure 5-1-4

(5) After the shared file is displayed, select and open the G code file that has been

set to be shared.

Figure 5-1-5

(6) 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.

(7) 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 5-1-6).
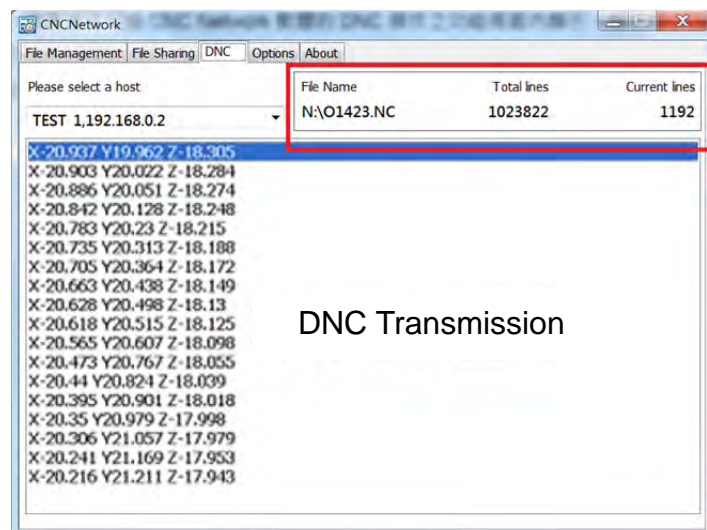
Figure 5-1-6

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

(3) In the screen of [File manage], press **↑;** **↓,** or **PAGE UP** **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 **F1** key (NEW file) and the dialog box for file name will pop up.

PROGRAM(File manage)	G42-G01-G	N1	SFT

CF:	Size	Date	Time
08020.NC	20 MB	2011/06/28	13:24
08010.NC	10 MB	2008/03/31	01:43
011	1 MB	2008/01/16	20:39
02001	78 B	2013/01/11	17:55
013.NC		013/04/03	17:32
00009		008/01/28	08:43
01042		012/07/24	13:57
00008.NC		008/04/21	23:36
01425.NC	24 MB	2008/02/27	23:11
00005	112 B	2012/07/31	17:03
00010	57 B	2013/04/23	10:05
01423.NC	24 MB	2008/02/25	06:18
00819.NC	497 KB	2008/06/30	04:51

NEW FILE

Input new file name: [   ]

EDIT			Ready

Figure 5-2-1

(5) Type alphanumeric letters (symbols are not included) in the box and press **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 subroutines)
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
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:
   (a) File name must be unique in one directory, e.g. O0001 and O1 are regarded as the same.
   (b) Only the machining files are displayed in the screen of [File manage]. The macro files display only upon special permission.

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

(3) In the screen of [File manage], press ↑; ↓, or PAGE UP 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 **F2** key (Copy file) to copy the file. Please note that it is required to execute the paste function to create a target file.

5.4   Paste

As described in Section **5.3**, 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 **5.3**).

(5) Use [↑], [↓], keys or [PAGE UP] [PAGE DN] keys to move the cursor to the disk, data directory or layer of the target file.

(6) Enter the directory of the target file and press [F3] key (Paste). Then, enter a new name or use the old name of the target file in the popup dialog box. Press [ENTER] key and file coping and pasting is done.

Note:
  (a) Please note that if the newly copied file exists in the same directory, then its name must differ from the source one.
  (b) 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.

5.5   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 [PRG] key to switch to the screen of [PROGRAM].

  (3) Press [↑], [↓], keys or [PAGE UP] [PAGE DN] keys to move the cursor and press [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 [F4] key (DEL) and the "Do you really want to delete?" dialog box will pop up. Press "Y" and [ENTER] key to delete the selected file or folder.

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

5.6   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:

(1) Set system to **Edit mode**.

(2) Press **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 ↑; ↓, keys or PAGE UP PAGE DN keys to move the cursor to the desired files. Press **F5** key (SEL TOGL) to select the file (see Figure 5-6-1). For files that have been selected, pressing the **F5** key (SEL TOGL) again will cancel the selection.
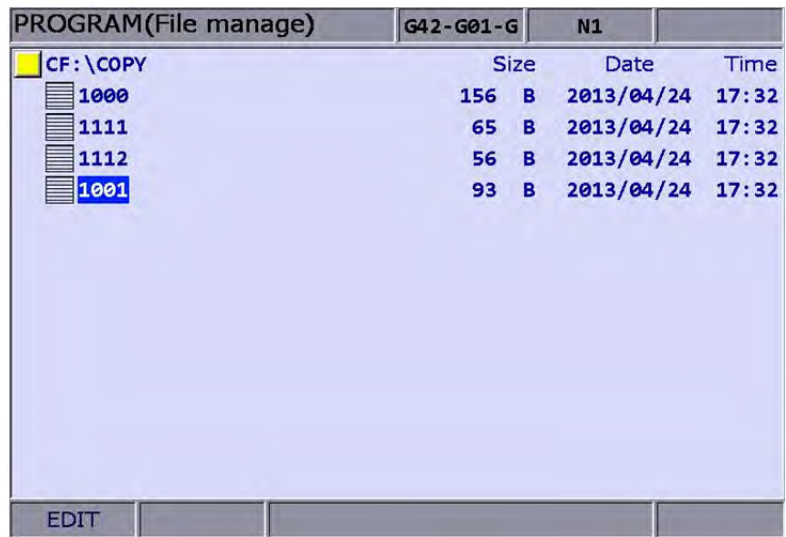
PROGRAM(File manage)	G42-G01-G	N1	
CF:\POSITION	Size	Date	Time
05405-1	1019 B	2012/07/20	10:07
05405-2	330 B	2012/07/20	10:13
05406-1	164 B	2012/07/24	09:03
05406-2	164 B	2012/07/24	09:09
1000	156 B	2012/08/30	14:35
1111	65 B	2012/09/10	16:48
1112	56 B	2012/07/10	16:45
1113	65 B	2012/08/10	17:23
1001	93 B	2012/09/04	15:07
5401-01	303 B	2012/11/02	13:35
G54-G28	499 B	2012/07/09	14:30
G54-G28-1	753 B	2012/06/29	15:39
05401	286 B	2012/06/26	08:53
EDIT			

Figure 5-6-1

(5) Press **F2** key (Copy file) to copy multiple files.

(6) Move the cursor to another directory. Press [F3] key (Paste) to paste multiple files as shown in Figure 5-6-2

PROGRAM(File manage)	G42-G01-G	N1	

CF:\COPY	Size	Date	Time
1000	156 B	2013/04/24	17:32
1111	65 B	2013/04/24	17:32
1112	56 B	2013/04/24	17:32
1001	93 B	2013/04/24	17:32

EDIT			

Figure 5-6-2

See the operation steps described below:

(1) Set the system to **Edit mode**.

(2) Press [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 [↑], [↓], keys or [PAGE UP] [PAGE DN] keys to move the cursor to the desired files. Press [F5] key (SEL TOGL) to select the file(see Figure 5-6-1). For files that have been selected, pressing the [F5] key (SEL TOGL) again will cancel the selection.

(5) Press [F4] key (DEL) and the dialog box for confirmation will pop up (see Figure 5-6-3). Press "Y" and [ENTER] key to delete the selected files.
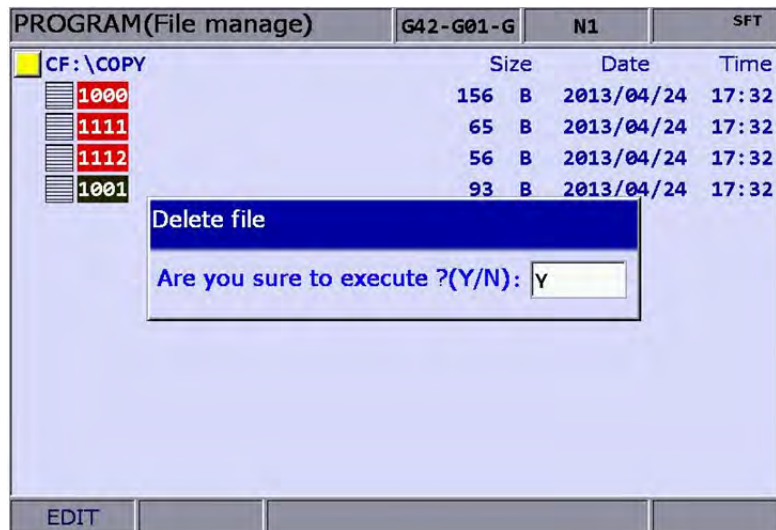
Figure 5-6-3

Note:

    (a) 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.

    (b) When there are duplicated file names while copying multiple files, the NC200 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.

5.7   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 [PRG] key to switch to the screen of [PROGRAM].
>
> (3) In the screen of [File manage], press [↑], [↓], or [PAGE UP] [PAGE DN] keys to move the cursor to the destination (e.g. the 2nd or 3rd layer in the CF or USB directory).
>
> (4) Press [►] key to switch to the function bar at next page.
>
> (5) Move the cursor to the file that you wish to rename. Then, press [F2] key (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 [ENTER] key.

>   **Note:**
>> (a) A G code file can be added in layer two or three but not layer one in [File manage].
>> (b) 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.

5.8   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 directory folders and G code files. See the operation steps below:

(1)   Set the system to **Edit mode**.

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

(3)   Press **▶** key to switch to the function bar at next page.

(4)   Press **F1** key (FOLDER) in the second layer of the [File manage], the dialog box for entering directory name will pop up.

PROGRAM(File manage)	G42-G01-G	N1		SFT

CF:	Size	Date	Time
MOTION		2011/09/15	14:05
MACRO TEST		2012/08/01	15:25
O_MACRO		2008/01/01	00:00
POSITION		2012/06/26	08:49
TEST		10/08/06	13:44
TOTAL-TEST		12/08/13	10:02
TEST-1		12/07/05	14:14
TEACH-TEST		13/04/24	11:40
TEST-0910		2012/09/10	13:58
COPY		2013/04/24	17:32
11111	128 B	2012/06/25	13:12
222.NC	86 KB	2012/02/22	11:03
6-1.ANC	36 KB	2012/10/26	08:44

Create directory

Input directory name: 123

EDIT

Figure 5-8-1

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

This creates a new directory in the second layer of [File manage]. Users now can do file creation and editing of 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

5.9 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 [PRG] key to switch to screen of [PROGRAM].

(3) In the screen of [File manage], press [↑] , [↓] , keys or [PAGE UP] [PAGE DN] keys

to move the cursor and press [ENTER] key to go to the destination in the second or third layer in the disk.

(4) Press [▶] key to switch to the function bar at next page.

(5) Press [F3] key (FIND FILE) and the dialog box will pop up. Enter the

desired file name in the box and press [ENTER] key to search and open the target file.

Note:
      (a) Instead of searching all directories, this file searching function is limited to one directory.
      (b) To find the correct file and open it, please enter the complete and exact file name you wish to search for.

5.10   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 [PRG] key to switch to screen of [PROGRAM].

(3)   In the screen of [File manage], press [↑]; [↓], keys or [▲ PAGE UP] [▼ PAGE DN] keys to move the cursor and press [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 [F2] key (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 [▶] key to switch to the function bar at next page.

(8)   Press [F4] key (MERGE) and the dialog box will pop up. Enter the desired file name and press [ENTER] key to open the target file.

(9)   Move the cursor to the location in the target file to paste the source file. Press [F2] key (Paste) 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.

5.11   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 5.12**. 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.**

5.12   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 the first 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 ▨ PRG key to switch to screen of [PROGRAM].

(3) Press ▨ ▨ keys or ▨ PAGE UP ▨ PAGE DN keys to move the cursor and press ▨ 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 ▨ ENTER key to open the file and enter edit mode.

(5) Use ▨ ▨ ▨ ▨ keys to move the cursor to any location in the file.

(6) Press the text, number, and edit keys in the first 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
Supported mode	Edit mode
Size limit of editable files	Only the file with size less than 3 MB can be edited.

Note: 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.

5.12.1   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 **PRG** key to switch to screen of [PROGRAM].

(3) Press ⬆, ⬇, keys or **PAGE UP** **PAGE DN** keys to move the cursor and press **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 **ENTER** key to open the file and enter the screen for file editing.

(5) Press ▶ key to switch to the function bar at next page.

(6) Press **F1** key (LABEL) and the dialog box for entering line number (key pad 0 ~ 9) will pop up.

(7) Enter the desired line number and press **ENTER** key. The will cursor move to the given line number and finish searching.

Searching rules of Line number

Searching rules of Line number	
Maximum length of searching string	62
Format of searching	Actual line number of program (key pad 0 ~ 9)

5.12.2 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 **PRG** key to switch to screen of [PROGRAM].

(3) Press **↑**, **↓**, keys or **PAGE UP** **PAGE DN** keys to move the cursor and press **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 **ENTER** key to open the file and enter the screen for file editing.

(5) Press **▶** key to switch to the function bar at next page.

(6) Press **F2** key (STRING) and the dialog box for entering the string will pop up.

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

(8) The string will be highlighted and the 'Forward' and 'Backward' options are displayed in the function bar.

(9) Repeatedly press **F1** key (NEXT) to search the next match. Press **F2** key (PREV) to search the previous match.

(10) Press **◀** key to exit the string search page. The function bar resumes displaying options of file editing.

(11) To find different key words, return to the edit function bar and repeat Step (6) ~ (9).

Keyword searching rule:

Keyword searching rule	
Maximum length of keyword	63

5.12.3   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 beginning 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 ![PRG] key to switch to screen of [PROGRAM].

(3) Press ![up]; ![down], keys or ![PAGE UP] ![PAGE DN] keys to move the cursor and press ![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 ![ENTER] key to open the file and enter the screen for file editing.

(5) Press ![F5] key (B start) to set the current cursor position as the starting point of the block.

(6) Use ![up]; ![down], ![left]&, ![right]# keys to move the cursor to position as desired block end.

(7) Press ![F6] key (B end) to set the current cursor position as the end of the block. See the figure below.

PROGRAM(File edit)	037	N25	SFT

```
N90  G01 X 0.33179 Y 0.66909
N100 G01 X 0.36931 Y 0.72325
N110 G01 X 0.41432 Y 0.76741
N120 G01 X 0.46851 Y 0.80741
N130 G01 X 0.52770 Y 0.83990
N140 G01 X 0.55354 Y 0.85157
N150 G01 X 0.58189 Y 0.86157
N160 G01 X 0.61023 Y 0.87073
N170 G01 X 0.63858 Y 0.87990
N180 G01 X 0.66692 Y 0.88990
N190 G01 X 0.69527 Y 0.90156
N200 G01 X 0.71861 Y 0.91573
N210 G01 X 0.73945 Y 0.93406
N220 G01 X 0.76113 Y 0.96239
N230 G01 X 0.77030 Y 0.99072
N240 G01 X 0.76780 Y 1.01821
N250 G01 X 0.75862 Y 1.04404
```

EDIT				Ready

Figure 5-12-1

(8) Follow Step (5) ~ (7) and press ![F3] key (DEL) to delete the text in the given block.

(9) Follow Step (5) ~ (7) and press [F1] key (Copy) to copy text in this block. Then,

move the cursor to the desired area for pasting and press [F2] key (Paste) to paste the selected text.

5.12.4   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 start and end point.

See the steps described below:

(1) Set the system to **Edit mode**.

(2) Press [PRG] key to switch to screen of [PROGRAM].

(3) Press [↑], [↓], keys or [PAGE UP] [PAGE DN] keys to move the cursor and press [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 [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 [F3] key (DEL) to delete the entire line.

(6) Delete a program block in the same way as described in **Section 5.12.3.** See Step (8) for defining the starting and ending points of a block.

5.12.5   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 [PRG] key to switch to screen of [PROGRAM].

(3) Press [↑], [↓], keys or [PAGE UP] [PAGE DN] keys to move the cursor and

press [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 [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 [F1] key (Copy).

(6) Move the cursor to the target location and press [F2] key (Paste) to paste the line.

(7) The entire block can be copied as described in **Section 5.12.3.** See step (9) for defining the starting and ending points of a block.

5.12.6 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 [PRG] key to switch to screen of [PROGRAM].

(3) Press [↑], [↓], keys or [PAGE UP] [PAGE DN] keys to move the cursor and

press [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 [ENTER] key to open the file and enter the screen for file editing.

(5) Press [F4] key (Undo) to undo the last action.

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

PROGRAM(Program execute)	037	N1	
O0037			

```
O0037
N10 G00 G57 Z 0.25000
N20 G00 X 0.36264 Y 0.34746
N30 G01 Z-0.10000 F 200.
N40 G01 X 0.33430 Y 0.37996 F500.00
N50 G01 X 0.31345 Y 0.41995
N60 G01 X 0.29428 Y 0.48578
N70 G01 X 0.29178 Y 0.55160
N80 G01 X 0.30345 Y 0.61243
N90 G01 X 0.33179 Y 0.66909
N100 G01 X 0.36931 Y 0.72325
N110 G01 X 0.41432 Y 0.76741
N120 G01 X 0.46851 Y 0.80741
N130 G01 X 0.52770 Y 0.83990
N140 G01 X 0.55354 Y 0.85157
N150 G01 X 0.58189 Y 0.86157
N160 G01 X 0.61023 Y 0.87073
```

AUTO		RPD 100%	F 30%	S 101%	

Figure 5-13-1

(2) Press [PRG] key again to switch to the screen which displays both program and coordinates . See Figure below.
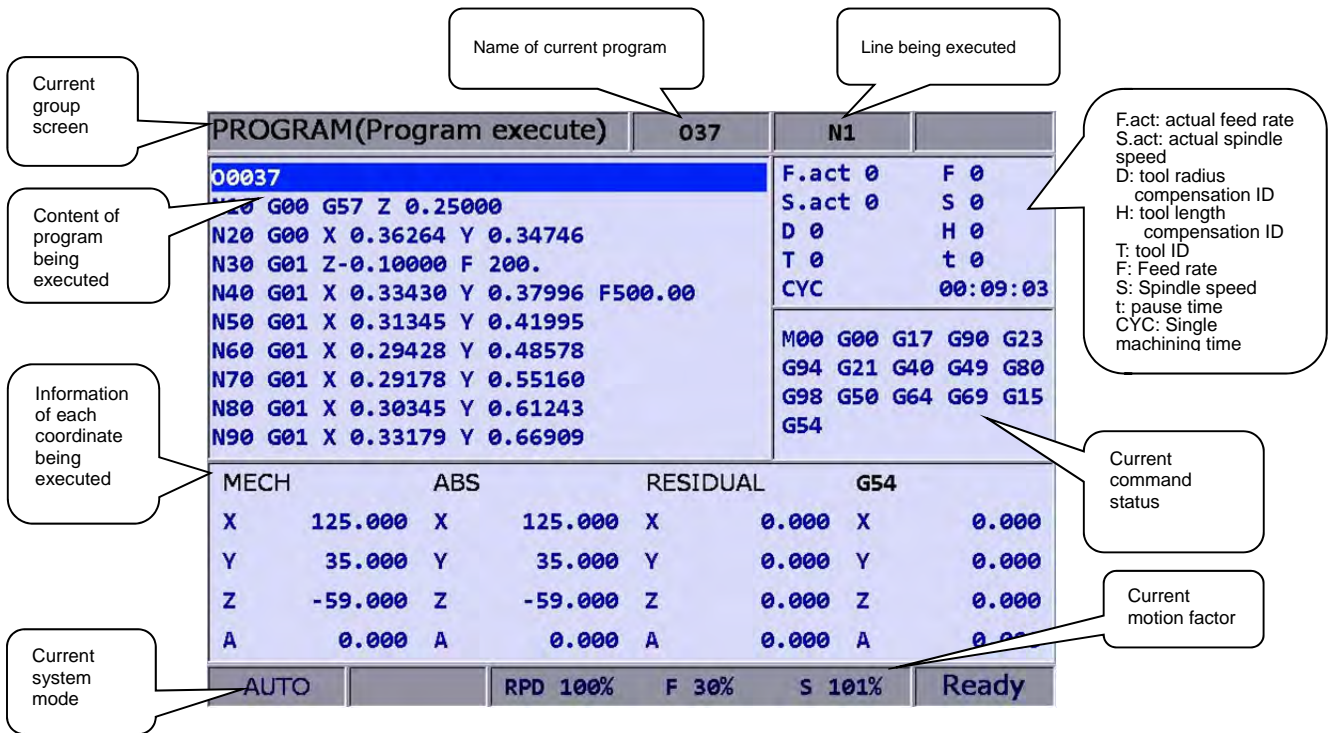
Figure 5-13-2

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 break point. See Figure below for illustration.
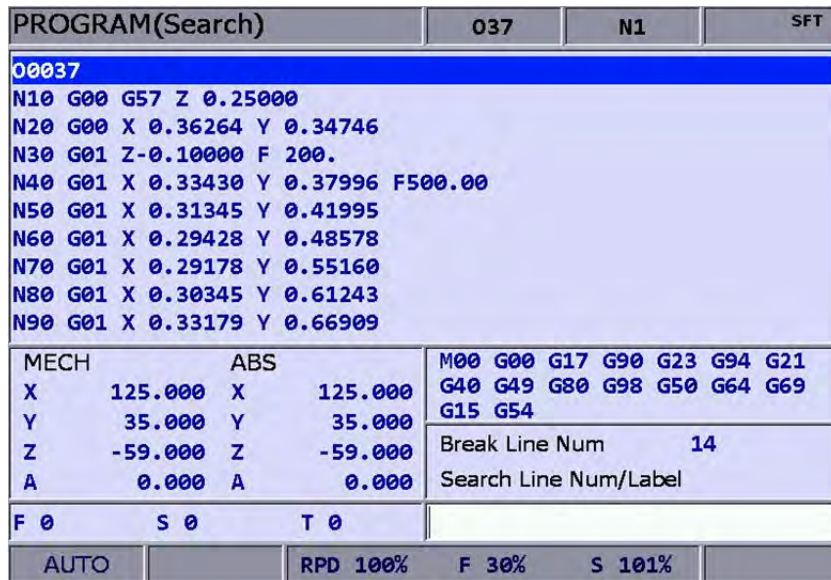
Figure 5-13-3

See the operation steps below:

(1)   In **Auto Mode**, press **PRG** key to enter the screen of [PROGRAM].

(2) Press **F2** key (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 **ENTER** key to complete the setting.

(4) Press **F1** key (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:

(a) 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.

(b) Valid search formats are the line number and N number of the program.

(c) 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 5-13-4 below. By using SF setup function and entering the new command value, the speed command is changed when program running.

Figure 5-13-4

See the operation steps below:

(1) In **Auto Mode**, press [PRG] key to enter the screen of [PROGRAM].

(2) Press [F1] key (SF set) and the dialog box for entering SF command will pop up.

(3) Enter the new S or F values then press [ENTER] key and the speed is changed.

Note:

(a) 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.

(b) 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.

(c) Do not use this function to change the existing speed command for a G code program without applying S and F commands.

JOG and hand wheel mode (JOG, MPG):

See the SF setting steps below:

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

(2) Press [F1] key (SF set) and the dialog box for SF command input will pop up.

(3) Enter new S or F values. Then, press [ENTER] key and the speed is changed.

Manual mode (MDI):

The PRG group function 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 manual 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 5-13-6

Figure 5-13-7

The file **save** function saves the manual edit file in the current directory following the same naming rule described in **Section 5-2 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:** 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 RESET key for 3 seconds.

(This page is intentionally left blank.)

Chapter 6 Offset (OFS) group

The OFS group provides setup functions for workpiece setup, cutting tool's length or radius compensation, and variables of macros.

6.1 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 6-1-1

See the operation steps described below:

(1) Press **OFS** key to enter the screen of [Offset].

(2) Press **F1** key (Coord) to enter the screen of coordinate system setup function bar.

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

6.1.1 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.

All Clear: It clears all axis value of the current coordinate system to zero while all other coordinate systems' value remains unchanged.
See the operation steps below:

(1) Press **OFS** key to enter the screen of [Offset].

(2) Press **F1** key (Coord) to enter the screen of coordinate system setup function bar.

(3) Press **F1** key (Auto) to switch to the screen with the coordinates auto setup function bar.

(4) Use **↑;** **↓,** **←&** **→#** keys to move the cursor to the position of the specified coordinate system group.

(5) Press **F2** key (CLR ALL) to remove all coordinates group data where the cursor resides.

Relative clear: This function clears the relative coordinates that corresponds 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 [Hand wheel 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 **OFS** key to enter the screen of [Offset].

(3) Press **F1** key (Coord) to enter the screen of coordinate system setup function bar.

(4) Press **F1** key (Auto) to switch to the screen with the coordinates auto setup function bar.

(5) Use [↑;] [↓,] [←&] [→#] keys to move the cursor to the X-axis coordinates position of the specified coordinate system group.

(6) Press [F4] key (SET L/2) to enter the L/2 input screen.

(7) Press [F1] key (Point1). See Figure 6-1-2, the first circle on the rectangle turns into red; this means the recording of the first mechanical coordinate value is complete.

```
OFFSET(Set coord system)        3        N1          mm

OFFSET        G54          MECH        REL
X      0.000 X      0.000  X   -100.250 X   -100.250
Y      0.000 Y      0.000  Y      0.000 Y      0.000
Z      0.000 Z      0.000  Z      0.000 Z      0.000


G55          G56          G54
X      0.000 X     50.000          X
Y      0.000 Y     50.000
Z      0.000 Z     50.000  O        +        O


JOG                RPD 100%   JOG 5000   S 100%
```

Figure 6-1-2

(8) Move the mechanism to the contact point on the other side of the X-axis.

(9) Press [F2] key (Point2). See Figure 6-1-2.When the second circle turns red, it means the mechanical coordinate value of the second point has been recorded.

(10) Press [F3] key (SET). The system will figure out 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 coordinates only. See the operation steps described below:

(1) In **[Jog mode]** or **[Hand wheel mode]**, move the mechanism to the first contact point on the X-axis of the workpiece coordinates.

(2) Press [OFS] key to enter the screen of [Offset].

(3) Press [F1] key (Coord) to enter the screen of coordinate system setup function bar.

(4) Press **F1** key (Auto) to switch to the screen with the coordinates auto setup function bar.

(5) Use **↑** ; **↓** , **←** & **→** # keys to move the cursor to the designated coordinate group and coordinate value of the axis. e.g. the setup position of X-, Y-, or Z-axis.

(6) Press **F3** key (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 6-1-4.
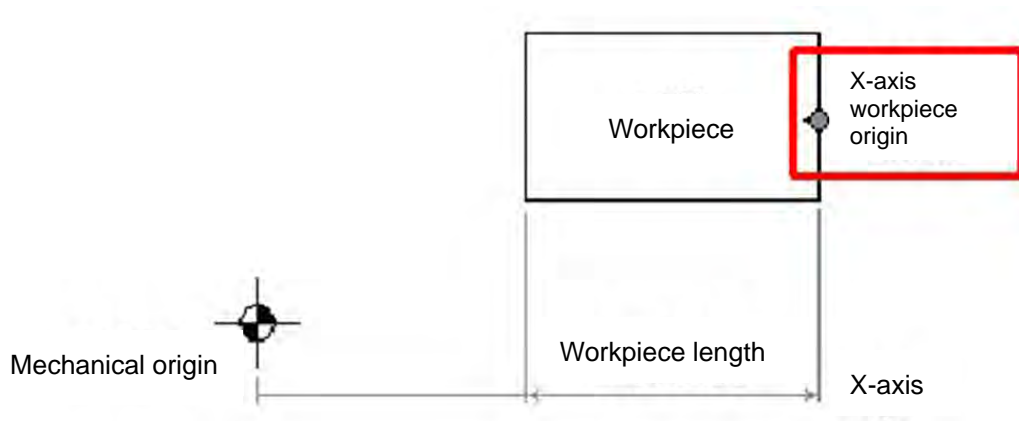
Figure 6-1-4

The mechanical coordinate data is shown in mechanical coordinate fields. Move the cursor to the specified coordinate group, such as G54 in Figure 6-1-5. Then, press **F3** key (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 6-1-5

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 [Hand wheel mode]**, move the mechanism to the initial contact point on the X-axis of the workpiece coordinates.

(2) Press ⬜ **OFS** key to enter the screen of [Offset].

(3) Press ⬜ **F1** key (Coord) to enter the screen of coordinate system setup function bar.

(4) Press ⬜ **F1** key (Auto) to switch to the screen with the coordinates auto setup function bar.

(5) Use ⬆; ⬇, ⬅& ➡# keys to move the cursor to the data position of the coordinate system group.

(6) Press ⬜ **F5** key (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 6-1-6. (Figure 6-1-6 indicates the relative position of the X- and Y-axis but not the Z-axis.)
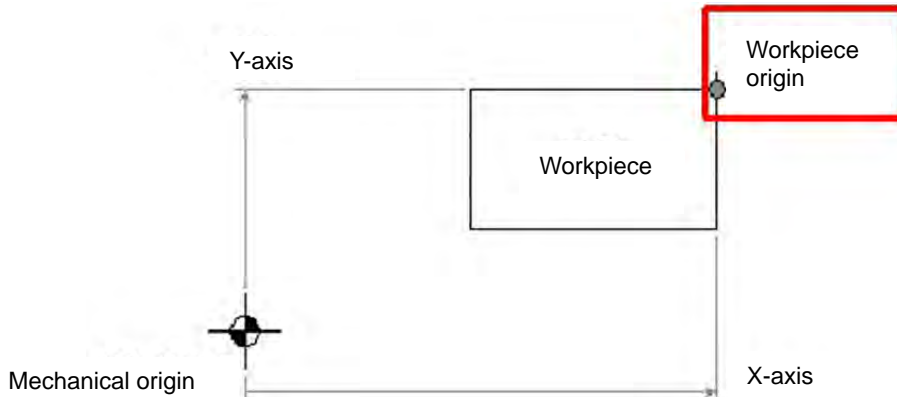
Figure 6-1-6

After the workpiece origin is calibrated, the mechanical coordinate data is shown in the mechanical coordinate fields in Figure 6-1-7. Move the cursor to the specified coordinates group (e.g. the G54 group in Figure 6-1-7). Press **F5** key (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 6-1-7

6.1.2 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 **OFS** key to enter the screen of [Offset].

(2) Press **F1** key (Coord) to enter the screen with coordinate system setup function bar.

(3) Use ↑; ↓, ←&, →# 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 **—]** key in advance. Press **·>** key to confirm the unit of values.

(5) Press **F2** key (ABS) to enter the value of the coordinate.

> **Note:**
> (a) The unit of value is mm. Value without decimal points is in unit of μm. That is, input value 123456 indicates 123.456 mm.
> (b) 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 6-1-8

Figure 6-1-9

6.1.3   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 is150.000, with an incremental input of 5.000, the new value shall be 155.000.

See the operation steps below:

(1) Press **OFS** key to enter the screen of [Offset].

(2) Press **F1** key (Coord) to enter the screen with coordinate system setup function bar.

(3) Use ⬆; ⬇, ⬅& ➡# 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 **—]** key in advance. Then, press **• >** key to confirm the unit of value.

(5) Press **F3** key (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.

6.1.4 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 6-1-10

See the operation steps below:
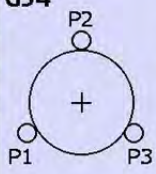
(1) Press [OFS] key to enter the screen of [Offset].

(2) Press [F1] key (Coord) to enter the screen with coordinate system setup function bar.

(3) Use [↑;] [↓,] [←&] [→#] keys to move the cursor to data fields of the specified coordinate system.

(4) Press [F4] key (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 [F1] [F2] [F3] [F4] (X1, X2, Y1, Y2) keys to set up the coordinates data of each point.

(6) Press [F5] key (Set) after coordinates of the four points are set, the system will figure out 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 [F6] key (SET Z) to set up the Z-axis coordinates of the workpiece coordinates group.

6.1.5   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:

OFFSET(Set coord system)		MDI	N4		
OFFSET	G54	MECH		REL	
X	0.000 X	0.000	X	60.000 X	-65.000
Y	0.000 Y	0.000	Y	80.000 Y	80.000
Z	0.000 Z	0.000	Z	-59.000 Z	-59.000
A	0.000 A	0.000	A	0.000 A	0.000
G55	G56	G54		P1	
X	150.000 X	56.000		X:	0.000
Y	100.000 Y	56.000		Y:	0.000
Z	0.000 Z	-56.000		P2 X:	0.000
A	0.000 A	0.000		Y:	0.000
				P3 X:	0.000
				Y:	0.000
MDI		RPD 100%	F 30%	S 101%	

Figure 6-1-13

See the operation steps below:

(1) Press **OFS** key to enter the screen of [Offset].

(2) Press **F1** key (Coord) to enter the screen with coordinate system setup function bar.

(3) Use **↑;** **↓,** **←&** **→#** keys to move the cursor to data fields of the specified coordinate system.

(4) Press **F5** key (CIRCLE) to enter the screen for circle center.

(5) Move the mechanical position of P1, P2, and P3 as guided. Then, press **F1** **F2** **F3** (P1, P2, P3) keys to set up the position one by one.

(6) After setting up coordinates of the three points on a circle, press **F4** key (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 **F6** key (SET Z) to set up the Z-axis coordinates of the workpiece coordinates group.

6.2 Tool register

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.

Figure 6-2-1

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 [OFS] key to enter the screen of [Offset].

(2) Press [F2] key (CUTTER) to enter the screen with the tool register function bar.

(3) Use [↑;] [↓,] [←&] [→#] 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 [—]] key in advance. Press [• >] 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 [F1] key (ABS) to register absolute values.

> **Note:** The tool data fields are for individual compensation values. For example, when the length fields are highlighted, then the input data are for tool length compensation values.

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 [OFS] key to enter the screen of [Offset].

(2) Press [F2] key (CUTTER) to enter the screen with the tool register function bar.

(3) Use [↑;] [↓,] [←&] [→#] 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 [—]] key in advance. Press [• >] 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 ▢F2 key (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 mode] or [Hand wheel mode],** move the Z-axis to specified coordinates height.

(2) Press ▢OFS key to enter the screen of [Offset].

(3) Press ▢F2 key (CUTTER) to enter the screen with the tool register function bar.

(4) Use ▲; ▼, ◄&, ►# keys to move the cursor to the tool length fields for tool code designation.

(5) Press ▢F3 key (SET H) to set the coordinates of the current Z-axis mechanical coordinates value to the given fields.

Note:
   (a)   The H setup function applies to tool length data fields only.
   (b)   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 fully not in operation, a block is completed when single-block stop function is enabled, and after the RESET key is pressed.
   (c)   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 ▢OFS key to enter the screen of [Offset].

(2) Press ▢F2 key (CUTTER) to enter the screen with tool register function bar.

(3) Press ▢F4 key (Clear) to display clear function bar.

(4) Press [F1] key (H/D) to clear tool length and radius values. Press [F2] key (Wear) to clear all tool length compensation and radius compensation values. Press [F3] key (Life) to clear all tool life span values. Press [F4] key (All) to clear all tool registry data.

6.3 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 can run in Jog mode only.** See the figure below.
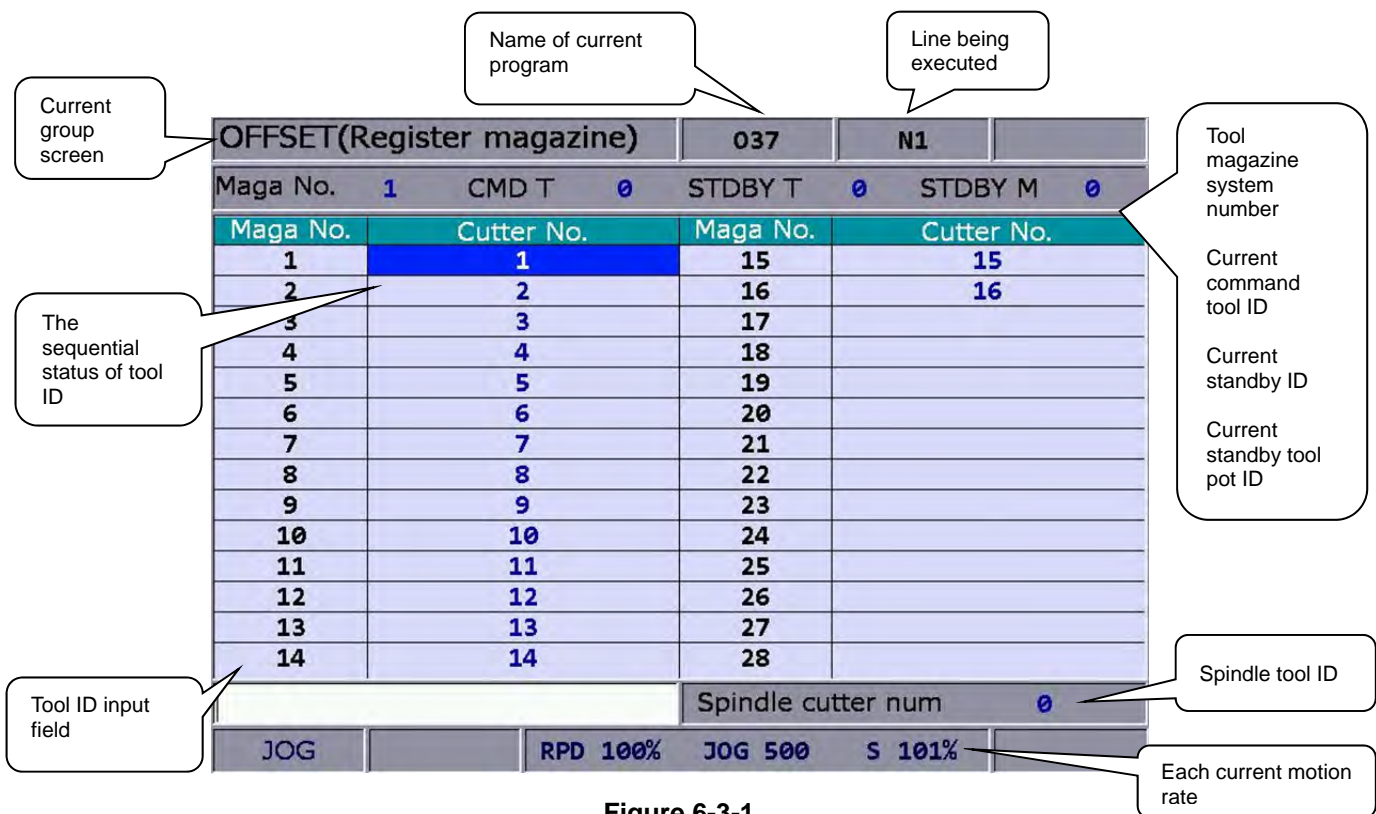
Figure 6-3-1

See the operation steps below for tool magazine setup:

(1) Set the system to **Jog mode**.

(2) Press [OFS] key to enter the screen of [Offset].

(3) Press [F3] key (MAGA) to enter the tool magazine data setup function screen.

(4) Use [↑;] [↓,] [←&] [→#] keys to move the cursor to assigned data fields.

(5) Enter the newly changed tool ID, press [F1] key (SET) (or press [ENTER] key) to set up the position of new tool magazine.

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 mode**.

(2) Press [OFS] key to enter the screen of [Offset].

(3) Press [F3] key (MAGA) to enter the screen of tool magazine data setup function.

(4) Press [F2] key (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 mode**.

(2) Press [OFS] key to enter the screen of [Offset].

(3) Press [F3] key (MAGA) to enter the screen of tool magazine data setup function.

(4) Use [↑;] [↓,] [←&] [→#] keys to move the cursor to the assigned data fields.

(5) Press [F3] key (LOCK) to block the tool magazine position as shown in Figure 6-3-5.

Figure 6-3-5

To unblock the tool magazine, continue with the steps below:

(6) In the tool magazine setup screen, press ⬆; ⬇, ⬅& ➡# keys to move the cursor to the blocked fields.

(7) Enter the specified tool ID; press **ENTER** to unblock the tool magazine position.

Note:

    (a) The tool magazine ID can be set in **Jog mode** (JOG) only. The tool magazine setup options will not be shown in other modes.

    (b) The special user permission is a must before doing tool magazine ID setup or reset.

    (c) 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.

    (d) 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.

6.4 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.

No.	Value	No.	Value
1	0.000	16	0.000
2	0.000	17	0.000
3	0.000	18	0.000
4	0.000	19	0.000
5	0.000	20	0.000
6	0.000	21	0.000
7	0.000	22	0.000
8	0.000	23	0.000
9	0.000	24	0.000
10	0.000	25	0.000
11	0.000	26	0.000
12	0.000	27	0.000
13	0.000	28	0.000
14	0.000	29	0.000
15	0.000	30	0.000

OFFSET(Macro var-local)　037　N1

JOG　RPD 100%　JOG 500　S 101%　STOP

Figure 6-4

6.4.1 Local variable

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

(1) Press [OFS] key to enter the screen of [Offset].

(2) Press [F4] key (Macro) to enter the screen of variable value entry.

(3) Press [F1] key (Local) to enter the entry screen for field ID starting with number 1.

(4) Use [↑;], [↓,], [←&], [→#] keys to move the cursor to the desired variable data field.

(5) Enter variable settings and press [ENTER] key to complete the setting.

6.4.2   Global variable

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

See the operation steps below:

(1) Press **OFS** key to enter the screen of [Offset].

(2) Press **F4** key (Macro) to enter the screen of variable value entry.

(3) Press **F2** key (Global) to enter the entry screen for field ID starting with number 51.

(4) Use ↑; ↓, ←& →# keys to move the cursor to the desired variable data field.

(5) Enter variable settings and press **ENTER** key to complete the setting.

6.4.3   Retaining variable

These variables retain system data after power outage. They are **numbered from 1601 ~ 1800.**

See the operation steps below:

(1) Press **OFS** key to enter the screen of [Offset].

(2) Press **F4** key (Macro) to enter the screen for variable value entry.

(3) Press **F3** key (Hold) to enter the entry screen for field ID started with number 1601.

(4) Use ↑; ↓, ←& →# keys to move the cursor to the desired variable data field.

(5) Enter variable settings and press **ENTER** key to complete the setting

Chapter 7 Diagnosis (DGN) Group

The DGN group provides machining information, user variable, system monitoring, and parameters import/export function to optimize the system.

7.1 Machining information (PROCESS)

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

DIAGNOSE(PROCESS)	00311	N213	

Total time 48563: 08: 01

Single time 13: 01: 42

Target stocks 150

Completed stocks 20

Date 2013/04/25   Time 08:38:56

AUTO   RPD 100%   F 30%   S 101%

Figure 7-1-1

See the operation steps below for machining information setup:

(1) Press **DGN** key to enter the page of [DIAGNOSE] .

(2) Press **F1** key (PROCESS) to enter the screen for machining information [PROCESS].

(3) Press **F1** key (Set NR), the machining count setup screen will pop up as shown in the figure below.

Figure 7-1-2

(4) Use ⬆; ⬇, keys to move the cursor to the specified field.

(5) Enter the value in the range of 0 ~ 9999 and press 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 DGN key to enter the screen of [DIAGNOSE]

(2) Press F1 key (PROCESS) to enter the setting page for machining information [PROCESS].

(3) Press F2 key (CLR TIME) and the confirmation dialog box will pop up.

(4) Press "Y" (Yes) key and press ENTER to reset the machining time of a single workpiece

See the operation steps below for clearing the machining count:

(1) Press DGN key to enter the page of [DIAGNOSE].

(2) Press F1 key (PROCESS) to enter the page for machining information [PROCESS].

(3) Press F3 key (CLR NR) and the confirmation dialog box will pop up.

(4) Press "Y" (Yes) key and press ENTER to reset the count of workpieces that have been machined.

7.2   User variable

This function enables 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 devices control.

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			

DIAGNOSE(User Variable)   00311   N312

AUTO   RPD 100%   F 30%   S 101%   RUN

Figure 7-2-1

See the operation steps below:

(1) Press **DGN** key to enter the page of [DIAGNOSE].

(2) Press **F2** key (User VAR) to enter the setting screen.

(3) Use **↑**, **↓**, or **PAGE UP** **PAGE DN** keys to move the cursor to the specified data field.

(4) Enter the desired register number (D512~D1023). Then, press **ENTER** key to display the register data.

(5) 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.

(6) Use **F1** (US DEC), **F2** (HEX), **F3** (S DEC) or **F4** (FLOAT) to select the display format.

(7) Move the cursor to the data field that requires to be deleted. Press **F5** key (DEL) to delete the data.

7.3   System monitoring

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

7.3.1 I/O monitoring

NC200 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 **DGN** key to enter the screen of [DIAGNOSE].

(2) Press **F4** key (SYS MONI) to enter the screen for system monitoring.

(3) Press **F2** key (I/O MONI) to enter the status monitoring screen for the I/O expansion module.

7.3.2   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 7-3-2. In the figure below, the servo of axes Y, Z, A and the spindle are all in Off status and only the X-axis remains in ON status.

Figure 7-3-2

See the operation steps below:

(1) Press [DGN] key to enter the screen of [DIAGNOSE].

(2) Press [F4] key (SYS MONI) to switch to the system monitoring screen.

(3) Press [F3] key (SERVO) to enter the servo monitoring screen.

7.4   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.

7.4.1   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 described below for changing user password (illustrated with User permission 1):

(1) Press [DGN] key to enter the screen of [DIAGNOSE].

(2) Press [F6] key (PWD) to display the password setup function bar.

(3) **If [User permission 1] is locked**, press [F3] key (U1 SCP) and the password entry dialog box for unlocking [User permission 1] will pop up.

(4) Enter valid password and Press [ENTER] key to unlock [User permission 1] and display relevant function items.

(5) Press [F1] key (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 [ENTER] key to complete the setting.

See the operation steps described below for unlocking the user account:

(1) Press [DGN] to enter the screen of [DIAGNOSE].

(2) Press [F6] key (PWD) to display the password setup function bar.

(3) **When [User permission 1] is locked**, press [F3] key (U1 SCP) and the password entry dialog box for unlocking [User permission 1] will pop up.

(4) Enter the valid password for [User permission 1]. Press [ENTER] key to unlock and display relevant function items.

See the operation steps below for locking up a user account:

(1) Press [DGN] key to enter the screen of [DIAGNOSE].

(2) Press [F6] key (PWD) to display the password setup function bar.

(3) **When [User permission 1] is unlocked**, press [F3] (U1 SCP) to display relevant function items.

(4) Press [F2] key (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.

7.5   Import

The system features a parameter import/export function for managing system parameters. Users can import correct parameters to recover the system and export the modified parameter files for backup. This function can only be used with proper permissions. It is more efficient to perform this function when troubleshooting the system with parameter errors.

See the operation steps below for parameter import:

(1) Press [DGN] key to enter the screen of [DIAGNOSE].

(2) Press [►] key to switch to the next function page.

(3) Press [F3] key (IMPORT) to display window of [FILE] (see Figure 7-5-1).

Use [▲]; [▼], keys to select the file path and press [ENTER] key to access the file.
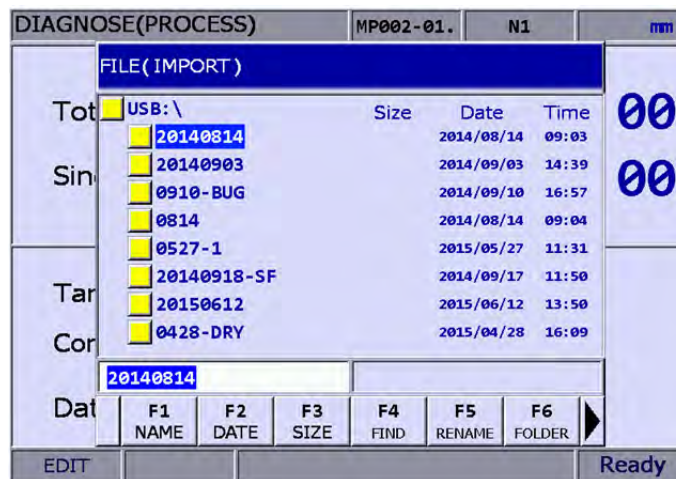
Figure 7-5-1

(4) Use [▲]; [▼], [◄&], [►#] keys to move the cursor. Press [ENTER] key to check the parameter items for import.

(5) Press [F2] key (SEL ALL) to select all the item or press [F3] key (CLR ALL) to cancel the selected items to be imported.

Figure 7-5-2

(6) Press [F1] key (IMPORT) to display the dialog box for confirmation. Enter "Y"

(yes) and press [ENTER], the data in the file will be imported to the system. Then, the importing progress will be displayed until it is complete.
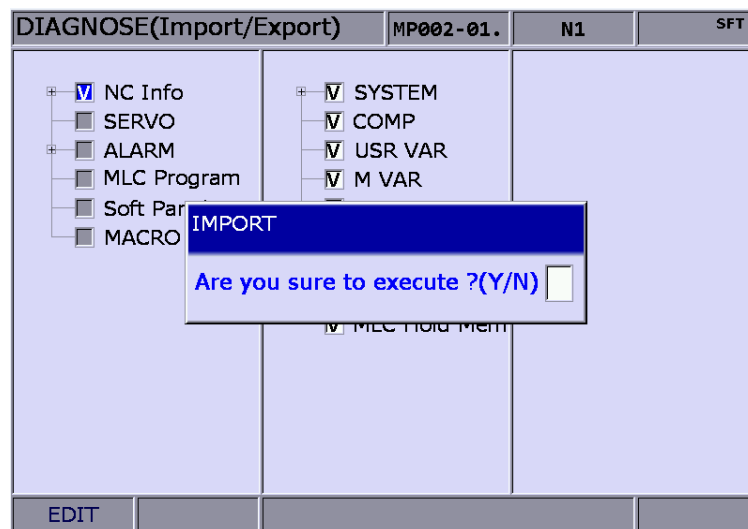
Figure 7-5-3

7.6   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. Please note that this function can be used only with proper permissions.

See the operation steps below for parameter export:

(1) Press [DGN] key to enter the screen of [DIAGNOSE].

(2) Press [►] key to switch to the next function page.

(3) Press [F4] key (EXPORT) to enter the screen of parameter export selection.

(4) Use [↑ ;], [↓ ,], [← &], [→ #] keys to move the cursor. Press [ENTER] key to check the items to export. Users may press [F2] key (SEL ALL) to select all items or press [F3] key (CLR ALL) to cancel all the sleceted items to export.

```
DIAGNOSE(Import/Export)    MP002-01.    N1         SFT

     ⊟─☑ NC Info          ⊞─☑ SYSTEM
        ☑ SERVO              ☑ COMP
     ⊞─☑ ALARM              ☑ USR VAR
        ☑ MLC Program        ☑ M VAR
        ☑ Soft Panel         ☑ M INFO
        ☑ MACRO           ⊞─☑ MACRO
                             ☑ COORD
                             ☑ CUTTER
                           ─☑ MLC Hold Mem




     EDIT
```
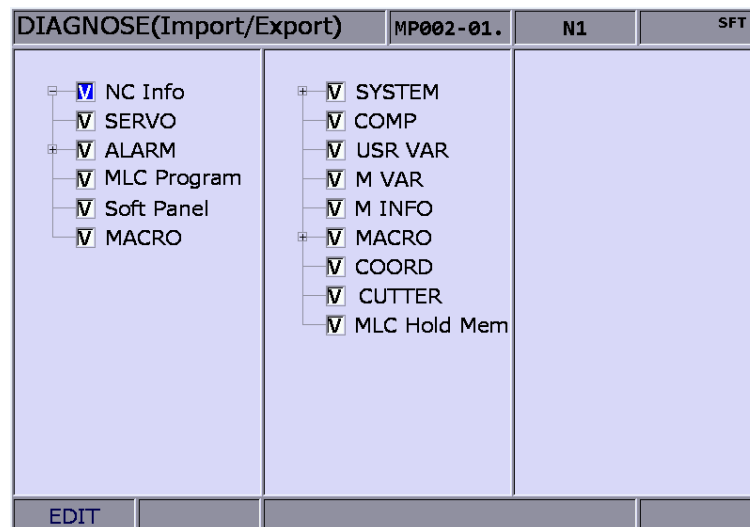
Figure 7-6-1

(5) Press [F1] key (EXPORT) to display the window of [FILE] (see Figure 7-6-2). Use [↑ ;], [↓ ,] keys to select file path of the saving destination or directly enter the file path in the directory. Then, press [ENTER] to save the exported data in the specified data file.
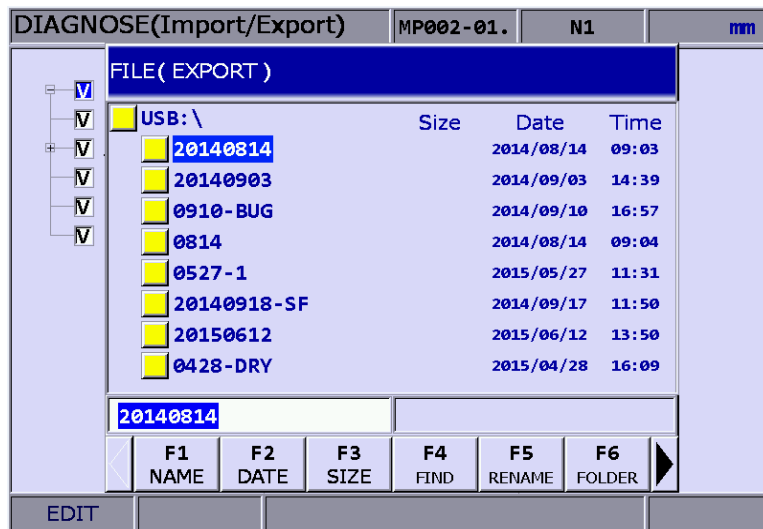
Figure 7-6-2

(6) After confirmed, the exporting progress will be shown until it is complete.

(7) To create a new file and save the exported data to this file (see Figure 7-6-2), please name this file and then press [F6] key (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 [ENTER] key to replace the existing data file.
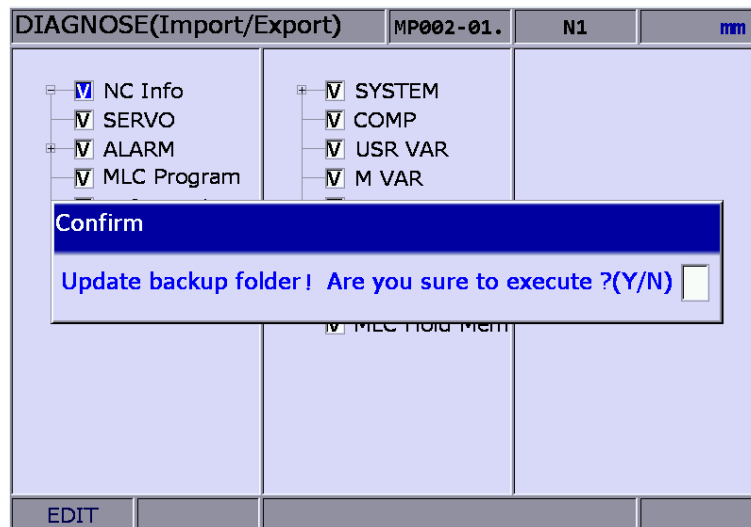
Figure 7-6-3

(9) In addition, the file management function [FILE] can be operated by related function keys (F1 ~ F6).

Chapter 8 Alarm (ALM) Group

The system prompts an alarm message for any program execution or command format error. The ALM group function displays current alarms sent by the system for troubleshooting.

8.1 Alarm

It is required to troubleshoot the issue that caused the alarm then press the **RESET** key to reset the system back to initial status. See the figure below for the alarm display screen with marked alert area.
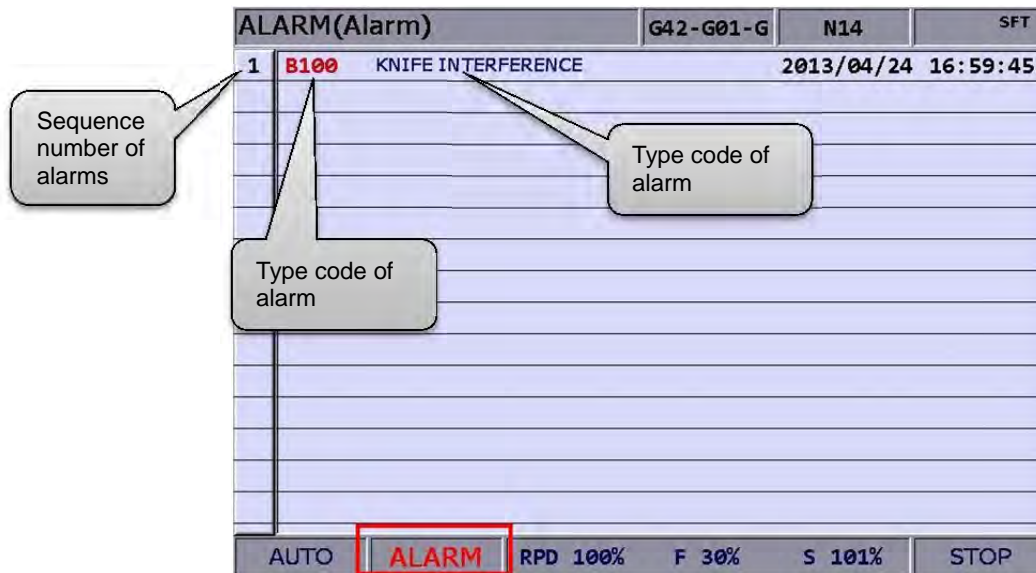
Figure 8-1-1

See the steps described below for the alarm message display and clear:

(1) Press [ALM] key to enter the screen of [Alarm].

(2) Press [F1] key (Alarm) to enter the current alarm message screen.

(3) Use [RESET] key to clear the alarm message now shown on the screen.

8.2   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 8-2-1

See the steps described below to clear all alarm history:

(1) Press **ALM** key to enter the screen of [Alarm].

(2) Press **F2** key (History) to enter the alarm history page.

(3) Press **F1** key (CLR ALL) and the confirmation dialog box will pop up.

(4) Press "Y" (yes) key then press **ENTER** key and all alarm records will be removed.

Note: The system switches to ALM group screen whenever an error occurs. Users can also set up parameter 10016 (The popup screen when an alarm occurs). The system will not switch to ALM screen automatically when an alarm occurs.

Chapter 9 Graphic (GRA) Group

The GRA group function displays trails during program running to help in ensuring correctness of the G code operation. 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.
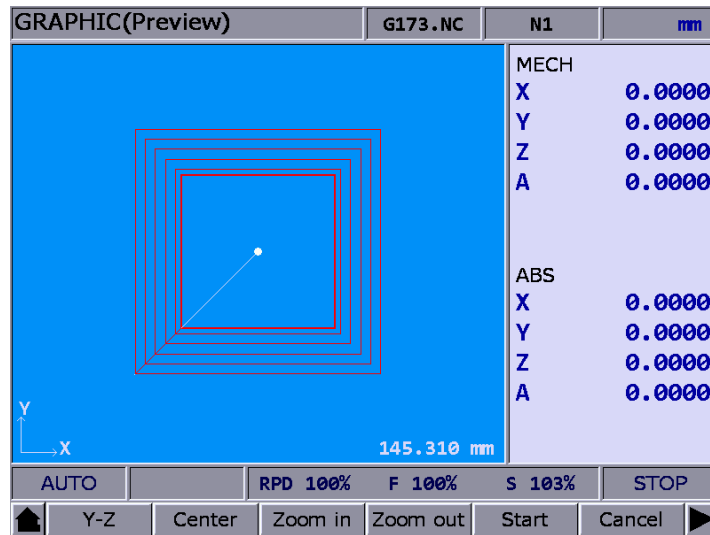
Figure 9-0-1

9.1 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 it executes machining program. 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 **GRA** key to enter screen of [GRAPHIC].

(2) Press **F1** key (CUTTING PATH) to the graphic display function.

(3) Press **F1** key (X—Y) to display the movement trail for plane of X-Y; or press **F1** key again to for plane of Y-Z ; or press **F1** key again to display the plane of X-Z; or press **F1** key again for plane of X-Y-Z.

(4) When machining program is running, entering GRA group will start the drawing.

Press [F6] key (STOP DRAW) to stop the drawing funciton of machining path.

Press [F5] key (DRAW) to continue the drawing.

(5) Press [F2] key (CENTER) to move the current motion display to the center.

Press [F3] key (ZOOM IN) or [F4] (ZOOM OUT) to magnify or minimize the display.

(6) Press [F1] (UP), [F2] (DOWN), [F3] (LEFT) and [F4] (RIGHT) keys (on the function bar at next page) to move the graph.

9.2 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 9.1. 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 steps below:

(1) Press [GRA] key to enter screen of [GRAPHIC].

(2) Press [F2] key (PREVIEW) to enter the screen for machining preview.

(3) Press [F1] key (X—Y) to display the movement trail for plane of X-Y; or press [F1] key again to for plane of Y-Z ; or press [F1] key again for plane of X-Z; or press [F1] key again for plane of X-Y-Z.

(4) Press [F5] key (PREVIEW) to view the machining result of G code file. Press [F6] key (CANCEL PREVIEW) to stop the preview.

(5) Press [F2] key (CENTER) to move the previewing graph to the center of the displaying frame. Press [F3] key (ZOOM IN) or [F4] (ZOOM OUT) to magnify or minimize the display.

(6) Press [F1] (UP), [F2] (DOWN), [F3] (LEFT) and [F4] (RIGHT) keys (on the function bar at next page) to move the graph.

Note:

    (a) When activating the display of machining path, function of machining preview cannot be enabled.

    (b) 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.

    (c) During machining preview, switching the mode will force the preview function to be cancelled.

    (d) If the preview has been cancelled, the next preview will start from the initial block when enabling again.

    (e) 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.)

Chapter 10 Parameter (PAR) Group

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.

10.1 General setting method

Generally, users have to enter new parameter values and then validate the setting based on the parameter types (S, P, and R). The parameter groups that apply general setting method are: operation, tool magazine, machining, spindle, mechanical, origin, compensation, and system.

(1) Press [PAR] key to enter the screen of [PARAMETER].

(2) Press [F1] key (Process) to enter to the screen for machining parameters.

(3) Use [↑]; [↓], keys to move the cursor to the specified data field. Enter the value within the proper range (please refer to the range displayed at the right down corner of the screen). See Figure 10-1-1. Then press [ENTER] to complete the setting.

PARAMETER(Process)		00311	N1	SFT
No.	**Parameter Name**			**Value**
307	Channel utility		P	197
	• Skip signal channel selection			0
	• Break point return			1
	• EMG stop mode			0
309	Nominal arc feed rate		R	1500
310	Minimal arc feed rate		R	500
311	Overlapped speed reduction ratio		R	50
315	F0 Speed		P	10
316	G00 Rapid speed		R	30000
317	G00 Rapid ACC/DEC time		R	200
318	Maximum moving speed		R	30000
319	ACC/DEC time		R	200
320	S curve time constant		R	20
321	ACC/DEC time		R	15
322	S curve time constant		R	5
		Range : 0 ~ 2		
JOG		Ch 0	1/3	

Figure 10-1-1

10.2 Alternative setting method

Apart from the setting steps described in Section 10.1, at the page of compensation parameter, it requires pressing the enter key to save the setting to the system. Please note this difference when setting compensation parameters.

(1) Press [PAR] key to enter the screen of [PARAMETER].

(2) Press [▶] key to switch to the screen for displaying the function bar.

(3) Press [F2] Key (Comp) to enter the screen for setting compensation parameters.

(4) Use [▲]; [▼], keys to move the cursor to the specified data field. Enter the value within the proper range (please refer to the range displayed at the right down corner of the screen). See Figure 10-2-1.

PARAMETER(Compensation)		00311	N1	
No.	Parameter Name	X	Y	Z
1000	Backlash amount R	0.00000	0.00000	0.00000
1001	Compensation time R	0	0	0
1002	Compensation delay time R	0	0	0
1003	Friction comp amount R	0.00000	0.00000	0.00000
1004	Friction comp time R	0	0	0
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	0.00000	0.00000	0.00000
1009	LSC Offset R	0.00000	0.00000	0.00000
		Range: -2 ~ 2 (mm, inch)		
JOG		Ch 0	1/10	

Figure 10-2-1

(5) Then press [ENTER] to complete the setting.

(6) The field of input length compensation can be completed by pressing [F5] key (um) to type in absolute length compensation in unit of um; or press [F6] key (um+) to type in incremental length compensation in unit of um.

(7) Compensation data generated by calibration instruments can be converted by the CNC SOFT program. Press [F1] key (import) to import compensation data which is the absolute type of data import; or press [F2] key (import +) to import compensation data as the incremental type of data import.

(8) After entering all compensation value, press [F1] key (OK) to confirm and update the compensation parameters.

10.3   Channel Setup (Ch/Axis Setting)

This function sets up the number and definition of the axis employed by the system as shown in Figure 10-3-1. The system mode cannot be set up in Auto or Manual modes.

Figure 10-3-1

See the operation steps below:

(1) Press [PAR] key to enter to the screen of [PARAMETER].

(2) Repeatedly press [►] key to switch to the screen with function bar.

(3) Press [F1] (Config) key to enter the screen for channel setup.

(4) Use [▲], [▼], keys to move to the fields of the specified axis function, press [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 [▲], [▼], [◄ &], [► #] keys to move the cursor and press [ENTER] key to check the desired field.

(6) After the axis attribute is defined, use [▲], [▼], [◄ &], [► #] keys to move the cursor to the port field of the axis, press [ENTER] key and the entry box for setting the up port number will pop up. Then, press [1 [] ~ [9 /] keys to enter a unique port number; press [ENTER] key and the port number of the axis is set.

(7) Press [F1] key (OK) after all axes are defined.
(8) Restart the NC200 control system to validate the setting.

Note:

    (a) 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. Either the NC or MLC axis can be selected, but not both, and assign a port number (unique from other axis port numbers).

    (b) To disable (cancel) the axis function, move the cursor to the specified field and press the ENTER key to uncheck the item and the function of this axis will be disabled.

10.4   RIO Setting

The NC200 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 10-4-1. See the operation steps below:
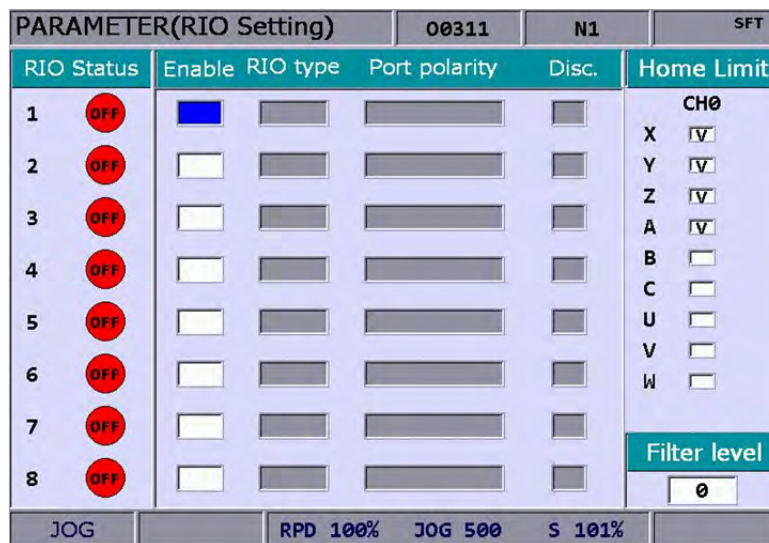
Figure 10-4-1

(1) Press [PAR] key to enter the screen of [PARAMETER].

(2) Repeatedly press [▶] key to switch to the screen with function bar.

(3) Press [F2] key (RIO) to enter the I/O expansion module setup page.

(4) Use [↑]; [↓], keys to move the cursor to the desired RIO port fields, press [ENTER] key to check the field and display the I/O setup screen of the selected port.

(5) Use [◀ &] [▶ #] keys to move the cursor to the polarity setup field, press [ENTER] key and the entry box will pop up. Then, press [ENTER] key after data entry and the polarity is set.

(6) Use [◀ &] [▶ #] keys to move to the field of [Disc.] (output when disconnected), press [ENTER] key to enable/disable the setting.

(7) Press [F1] key (OK) after all I/O modules function are set.

10.5   Search

This function enables users to search and point to desired parameter fields with a parameter code for faster and easier access to the screen containing a given parameter. See the operation steps described below:

(1) Press [PAR] key to enter the screen of [PARAMETER].

(2) Repeatedly press [▶] key to switch to the screen with function bar.

(3) Press [F3] key (Search) to start searching.

Note:

Apart from using the function key to search a 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.

Chapter 11 Table of G Codes

Table of supported G code for Milling machine

G code	Group	Function description
G00	01	Fast positioning command
G01	01	Linear cutting command
G02	01	CW Clockwise arc cutting (CW)
G03	01	Counterclockwise arc cutting (CCW)
G04	00	Pause command
G09	00	Exact stop
G10	00	Data entry setup
G11	00	Data entry setup cancelling
G15	16	Polar coordinates cancelling
G16	16	Polar coordinates
G17	02	X-Y plane selection
G18	02	Z-X plane selection
G19	02	Y-Z plane selection
G20	06	Inch input
G21	06	Metric input
G28	00	Homing through the reference origin point
G29	00	Homing to the starting point
G30	00	Auto homing of the second, third, and fourth reference point
G31	00	Skip function
G40	07	Tool radius compensation cancelling
G41	07	Tool radius left compensation
G42	07	Tool radius right compensation
G43	08	Tool length positive direction compensation
G44	08	Tool length negative direction compensation
G49	08	Tool length compensation cancelling
G50	11	Scale cutting cancelling
G51	11	Scale cutting
G52	00	Local coordinate system setup
G53	00	Mechanical coordinate system setup

G54	12	The first machining coordinate system selection
G55	12	The second machining coordinate system selection
G56	12	The third machining coordinate system selection
G57	12	The fourth machining coordinate system selection
G58	12	The fifth machining coordinate system selection
G59	12	The sixth machining coordinate system selection
G61	13	Exact stop mode
G64	13	Cutting mode
G65	00	Non-continuous effect macro command calling
G66	14	Continuous effect macro command calling
G67	14	Continuous effect macro command calling cancelling
G68	15	Coordinate system rotation command
G69	15	Coordinate system rotation command cancelling
G73	09	Peck drilling cycle
G74	09	Left spiral tapping cycle
G76	09	Fine boring cycle
G80	09	Constant loop cancelling
G81	09	Drilling cycle
G82	09	Countersunk drilling cycle
G83	09	Deep hole peck drilling cycle
G84	09	Right spiral tapping cycle
G85	09	Broaching cycle
G86	09	Rough boring cycle
G87	09	Rear boring cycle
G88	09	Boring cycle
G89	09	Boring cycle
G90	03	Absolute coordinate value system
G91	03	Incremental coordinate value system
G92	00	Coordinate system setup
G94	05	Feed rate setup (mm/min)
G98	10	Return to the initial point of the fixed cycle
G99	10	Return to the R point of the fixed cycle

Chapter 12 G Codes Description

G00: Fast positioning command

Format: G00 X_Y_Z_ (This command moves all three axes, any two or one of them as desired.)

    X_Y_Z_: Coordinates of the end point

Description: The G00 command quickly moves the center of the tool to the specified coordinate position (X, Y, Z). Its moving speed is tuned by the Fast Feed % key on the secondary control panel.

G01: Linear Cutting Command

Format: G01 X_Y_Z_F_

    X_Y_Z_: Coordinates of the end point.
    F_: Feed rate in unit of mm/min

Description: This command enables a cutter to linear cut from the current position to a given position at F feed rate. The feed rate is set by the F parameter in unit of mm/ min along with the **Feed Rate %** key of the secondary control panel.

G02/G03: Arc Cutting Command

Format: Arcs in the X – Y plane
    G17 G02 (or G03) X_ Y_ R_ F_ or
    G17 G02 (or G03) X_ Y_ I_ J_ F_
    You may add parameter Z_ in the command to spirally move in the X – Y plane.

    Arcs in the Z – X plane
    G18 G02 (or G03) Z_ X_ R_ F_ or
    G18 G02 (or G03) Z_ X_ K_ I_ F_
    You may add parameter Y_ in the command to spirally move in the Z – X plane.

    Arcs in the Y – Z plane
    G19 G02 (or G03) Y_ Z_ R_ F_ or
    G19 G02 (or G03) Y_ Z_ J_ K_ F_
    You may add parameter X_ in the command to spirally move in the Y – Z plane.

    G02: Clockwise (CW) arc cutting.
    G03: Counterclockwise (CCW) arc cutting.
    X, Y, and Z: Coordinates of the end point in absolute or incremental values determined by command G90 and G91.
    R: Arc radius (expressed in Radius format).
    I: The X-axis distance between arc center and starting point. The increment amount from the starting point to arc center.

J: The Y-axis distance between arc center and starting point. The increment amount from the starting point to arc center.

K: The Z-axis distance between arc center and starting point. The increment amount from the starting point to arc center.

(The I, J, and K expression is also known as the center method.)

F: The cutting feed rate in unit of mm/min.

Description: G02 or G03 is the arc cutting command. The arc cutting direction (G02 or G03) for a three-dimensional workpiece in individual planes is shown in Figure 1. The cutting direction is defined by the right hand coordinate system. Look against the positive direction of the plane's vertical axis; G02 is for clockwise direction while G03 is for counterclockwise.

X-Y plane   Z-X plane   Y-Z plane
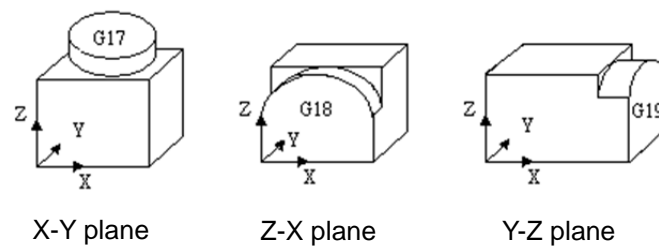
The center and radius methods are described below:

1. Radius method: R is the radius of the arc and shows in radius value. For the positive R value, it is an arc of central angle ≤ 180° and an arc of central angle > 180° for the negative R value.

β > 180° -> the B arc with negative R
α ≤ 180° -> the A arc with positive R
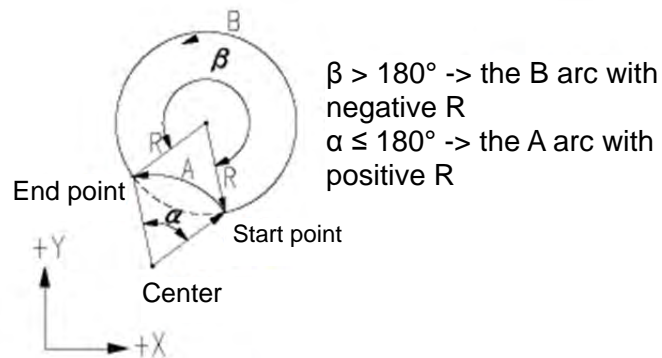
The Radius method

2. The Center method: Here parameters I, J, and K define the relative distance from the arc start point to the arc center (the end point). That is, the increments from the start point to the center in X, Y, and Z directions respectively. See the figure below for illustrations.
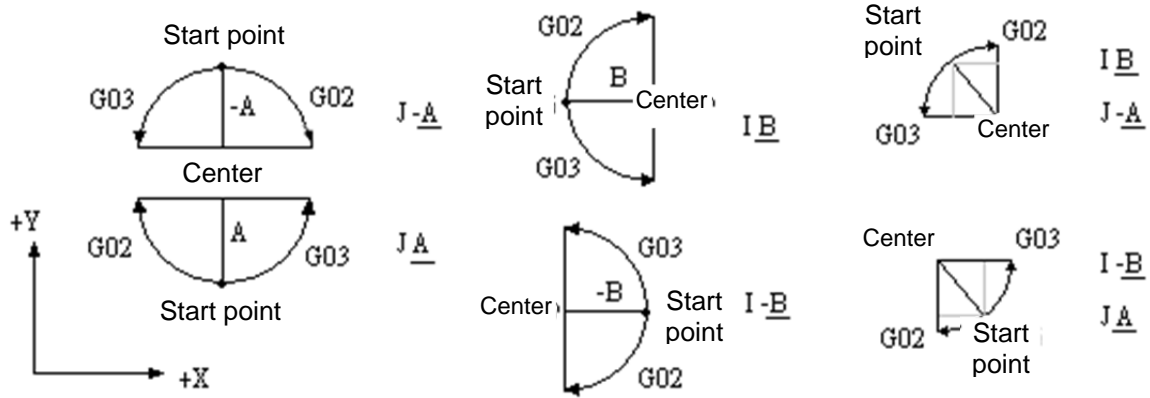
Values of parameter I and J / plus & minus sign on plane X-Y

Notes on G02 and G03 arc cutting:

(1) When I, J, and R parameters show in one statement only the R parameter remains active while I and J are ignored.

(2) Parameter I0, J0, and K0 can be omitted.

(3) When end point X, Y, and Z coordinates are omitted, it means the start and end points are the same and a sphere is to be cut and milled. The tool remains motionless for command in the radius method.

(4) The system prompts an alarm message when the end point does not intersect with the given radius value.

(5) For arc cutting following a linear cutting the G command must convert to command G02 or G03 and G01 for straight line cutting.

G04: Pause Command

Format: G04 X_ or
         G04 P_

Description: This command sets up a pause of the current node. Both **X** and **P** parameters
define the time of pause while **X** accepts decimal values and **P** does not.

Command scope:

Range of pause time by X parameter	
Range of valid values	Unit
0.001 ~ 99999.999	Seconds
Range of pause time by P parameter	
Range of valid values	Unit
1 ~ 99999999	0.001 second

G09：Exact Stop Command

Format: G09 G01 X_ Y_

Description: With G09 in effect, the system positions the tool every time it executes a
motion block. After confirming the tool is in the right position, the next block
can be executed. With command G09 used, there will be minor discontinuity
between blocks; this improves precision at the cost of speed. This command is
only active for cutting commands (G01 ~ G03) of a single block.

Tool movement
direction

Workpiece

With G09 in use

Without G09 in use

G10/G11: Data Entry Setup and Cancel

Format: G10 L2 P_ X_ Y_ Z_

> G10 L10 P_ R_: Tool length compensation
> G10 L11 P_ R_: Tool length wear compensation
> G10 L12 P_ R_: Tool radius compensation
> G10 L13 P_ R_: Tool radius wear compensation
> G10 L20 P_ X_ Y_ Z_: Expansive workpiece coordinate system entry

Description: The G10 command in the syntax of G10 L2 P_ X_ Y_ Z_ is used for workpiece coordinate system data entry. The system is set to the offset coordinate of the workpiece coordinate system when the value of parameter P is set to 0. Parameters P1 ~ P6 correspond to G54 ~ G59 of the workpiece coordinate system while X, Y, and Z represent the origin of the given coordinates system. The P parameter in L20 command syntax can assign the values of P1 ~ P64 for the corresponding expansion workpiece coordinates system. The command format G10 L10 P_ R_ sets up the tool length compensation value where parameter P is the compensation number and R is the actual compensation value for tool radius and length. Under the status specified by G90, the G10 command is absolute input. Data entry can also be done with command G91 and the input value of G10 will be increment.

Note:
  (1)   The G10 command is non-continuous and so it is effective in a single command block. The compensation values of the offset coordinates and job coordinates system are given relative to the origin of the mechanical coordinates system. You can execute command G11 to cancel data entry settings.

  (2)   During program execution, the command coordinates data changed by L2 or L20 command takes effects in next motion block. The tool compensation data changed by commands L10 ~ L13 take effect only after running compensation commands (G41/G42 or G43/G44) with compensation data number (D or H) again.

G15: Polar Coordinates Command Cancelling

Format: G15

Description: Command G15 cancels the status specified by the polar coordinates command.

G16: Polar Coordinates Command

Command syntax: G17 G16 X___ Y___ or;
G18 G16 Z___ X___ or;
G19 G16 Y___ Z___

X__ Y__: In a G17 plane the X__ parameter specifies the radius and Y__ the angle coordinates of the polar coordinates system.
Z__ X__: In a G18 plane the Z__ parameter specifies the radius and X__ the angle coordinates of the polar coordinates system.
Y__ Z__: In a G19 plane the Y__ parameter specifies the radius and Z__ the angle coordinates of the polar coordinates system.

Description: The polar coordinates command employs radius and angle as its setup format. If the first axis (X-axis) of the plane is selected for radius then the second one (Y-axis) is set for angle value. The counterclockwise angle is a positive value while the clockwise one is a negative value.

G17/G18/G19 : Plane Designation Command

Format: X - Y plane G17 {G01 ~ G03} X_ Y_{I_ J_ or R_}F_
Z - X plane G18 {G01 ~ G03} Z_ X_{K_ I_ or R_}F_
Y - Z plane G19 {G01 ~ G03} Y_ Z_{J_ K_ or R_}F_

Description: The plane selection function selects different planes for cutting. G17 ~ G19 commands set up the active plane for straight line and arc cutting or tool compensation. The system defaults to the G17 plane after power on.
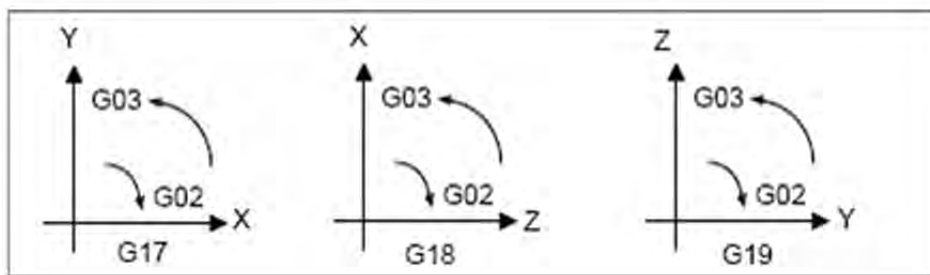
G21/G20: Metric / Inch Input Command

Format: G21 or G20

      G21: Set the metric system
      G20: Set the inch system

Description: This command sets the metric or inch unit of measure for the system.
      Command G21/G20 applies to the linear axis but not the rotation axis. It must
      be executed prior to the coordinate system setup command in a program.
      During the command execution, metric and inch input command cannot be
      switched. All relevant values which influences the system includes F value of
      cutting feed rate, coordinates position value, workpiece coordinates offset
      amount, tool compensation amount and movement distance. G21/G20 are
      continuous effective commands. When the program specifies a system unit, it
      means its unit is metric or inch unit. Moreover, G21 and G20 cannot be used in
      one program at the same time.

G28: Homing to the First Reference Point

Format: G90 G28 X_ Y_ Z_ or;
      G91 G28 X_ Y_ Z_

      X_ Y_ Z_: Coordinates of reference point

Description: This command instructs the tool to fast move (G00) from the reference point
      given by the command to the mechanical origin.
      The X_ Y_ Z_ of the format represents the coordinates of the reference point.
      The undesignated axis does not set up reference point and return to the origin.
      When G28 is executed, the tool radius compensation (G41 or G42) will be
      cancelled in advance. Thus, please set the tool radius compensation function
      again when executing the next block.

G29: Homing to the Starting Point

Format: G90 G29 X_ Y_ Z_ or;
      G91G29 X_ Y_ Z_

      X_ Y_ Z_: The final motion position of current block

Description: Command G29 moves a tool from the mechanical origin to a point defined by
      the command through a reference point. Command G29 and G28 must be
      used in pair. When running command G29 without accompanying G28 (for
      reference point setup), the system stops running and prompts an alarm
      message.

G30 : Auto Homing to the Second, Third, and Fourth Reference Point

Format: G30 P2 X‿ Y‿ Z‿ or ;
        G30 P3 X‿ Y‿ Z‿or ;
        G30 P4 X‿ Y‿ Z‿

      P‿: Select the reference point, the second, third or fourth
      X‿ Y‿ Z‿: Coordinates of intermediate point

Description: Parameters P2, P3, and P4 designate coordinates of reference points 2, 3 and 4. You can omit parameter P2 if the second reference point is required. The coordinates of the second, third, or fourth reference point are defined by the system parameters. This command is most commonly used for tool replacement. The tool radius and length compensation are cancelled automatically when executing command G28 and G30; thus, the compensation will not be functioning. The tool radius compensation function is resumed at the next block after G28 or G30.
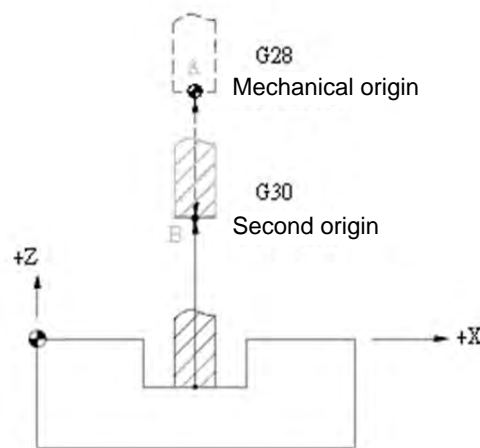
The example of command G30

G31: Skip Function Command

Format: G31   X‿ Y‿ Z‿ F‿

Description: During linear motion, G31 stops running the motion path immediately and execute the next block via external skipping signal. This command is only valid in single block. G31 cannot be executed when tool compensation is functioning. Thus, please cancel tool compensation before command execution.

G40: Tool Radius Compensation Cancelling Command

Format: G40 or
G40 X‗ Y‗

Description: This command cancels the tool compensation function when it is not needed in the tool path.

G41/G42: Tool Radius Left and Right Compensation Command

Format: G00 G90 G41 D‗ or;
G00 G90 G42 D‗

G41: Tool radius leftward compensation
G42: Tool radius rightward compensation
D‗: Code of tool radius compensation data

Description: For a program path without tool radius compensation, the tool cuts by profiling the workpiece shape with the tool center. That is, the effects of tool radius are not considered in the motion path. This leads to a machined workpiece with a size one tool diameter smaller than required. See the figure below.
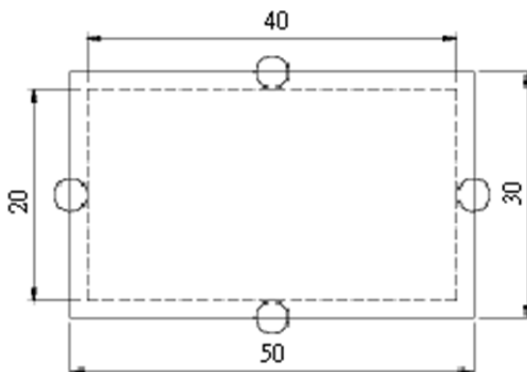
Tool radius: 10 mm
contour cutting without compensation
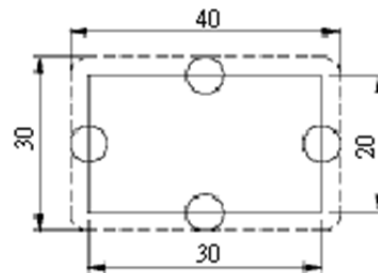
Tool radius: 10 mm
contour cutting without compensation

The leftward or rightward compensation of a tool is defined as below: Along the cutting direction, the tool radius shall be compensated rightward for tools moving to the right of the workpiece and made by command G42. Otherwise, the tool radius shall be compensated leftward for tools moving to the left of the workpiece and made by command G41.

D‗: This is the code of tool radius. This is the tool compensation data number contained in OFS group. For example: D11 indicates that the tool radius compensation number is 11. If number 11 represents value 4.0 then the tool radius is 4.0 mm.

Note for tool radius compensation:

   (1)   This command may be assigned together with G00 or G01 in the same block. The tool compensation is activated only after the tool moves (enable the tool radius compensation command). It cannot work together with commands G02 and G03. The compensating tool radius in the arc path must specify the compensation in advance in the straight line motion path. While the tool radius compensation is active, it cannot be cancelled in arc path. See below for illustration.

   (2)   When designing the program, the tool compensation number, e.g. D11 and D12, defined in a program must correspond to the one contained in the compensation table. These tool radius compensation values are numbers entered by the operator in the tool entry function of the OFS group in advance.

   (3)   When changing compensation value from positive to negative or vice versa, the compensation direction of G41 and G42 will be changed. For example, for positive arguments in command G41, the system compensates leftward and rightward when negative arguments are given. Similarly, for positive arguments in command G42, the system compensates rightward and leftward when negative arguments are given. That is, the function direction of the G41 and G42 exchange along with the change of the positive-negative sign changes in compensation value.

   (4)   The tool radius compensation function is deactivated temporarily when command G28 or G29 is in active status. The compensation status resumes when executing the next motion block as the compensation status is maintained by the control system.

   (5)   After the program path is completed under the tool radius compensation mode, the command G40 should be executed to cancel the compensation status and return the tool radius center back to its actual coordinate points. That is, after a G40 command is executed in a program, the motion path cancels the compensation length by turning the leftward or rightward compensation value to the opposite direction. The G40 command should be executed after the tool is moved away from the workpiece

   (6)   In compensation status, the linear moving distance and inner arc cutting radius must be greater than or equal to the tool radius; otherwise, over cutting may occur due to compensation vector interference. In a case like this, the controller stops operation and prompts alarm messages. Please see the figure below for the illustration of over cutting.
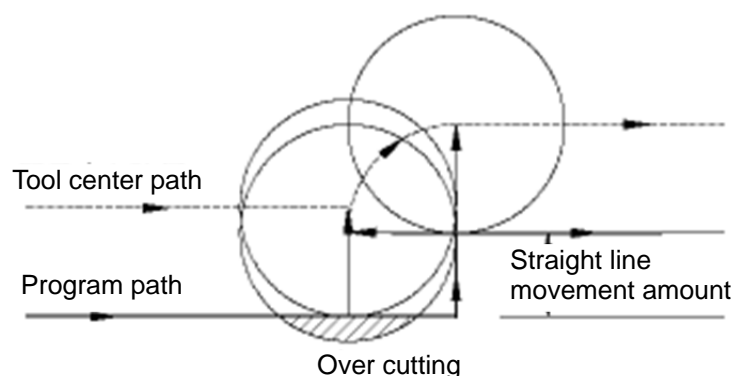
Straight line movement amount
is smaller than cutter radius

Tool radius compensation path type:

The G41/G42 tool radius compensation has two types. Type A does not compensate the starting and ending points in a path while Type B compensates both points. The compensation path must take angles (180° > θ> 90°, 0 < θ < 90°) formed by blocks into consideration. When degrees of angle formed by blocks are in the range of 180° and 90°, the tool moves in an inward curved angle and in an outward curved angle when the angle is in the range of 0° and 90°. See the illustration below for details.

Inward curved angle	Outward curved angle
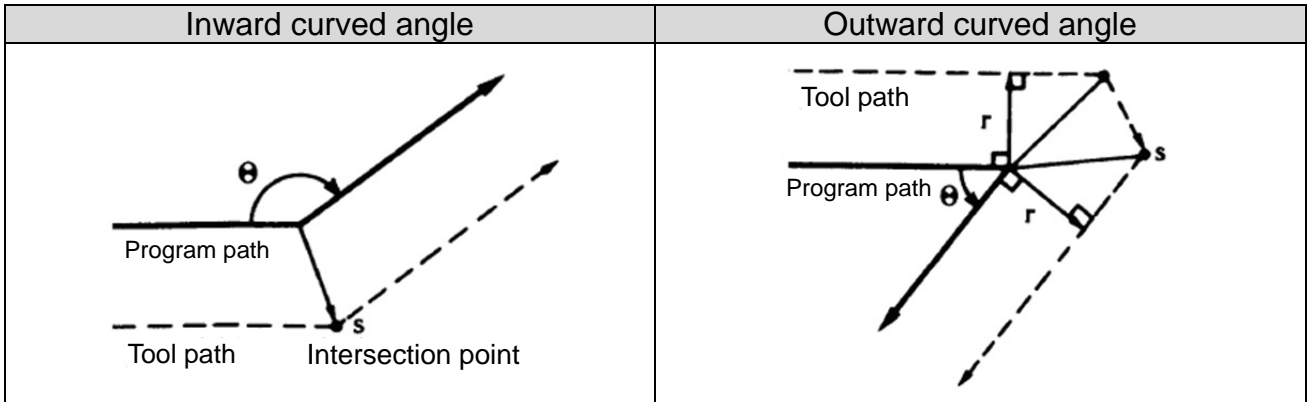
There are two compensation types: Type A and B. Type A does not compensate the starting and ending points in a path while Type B compensates both points. See the figure listed in the left for the motion tracks generated by Type A.

Type A: Line to line	Type B: Line to line
Type A: Line to arc	Type B: Line to arc
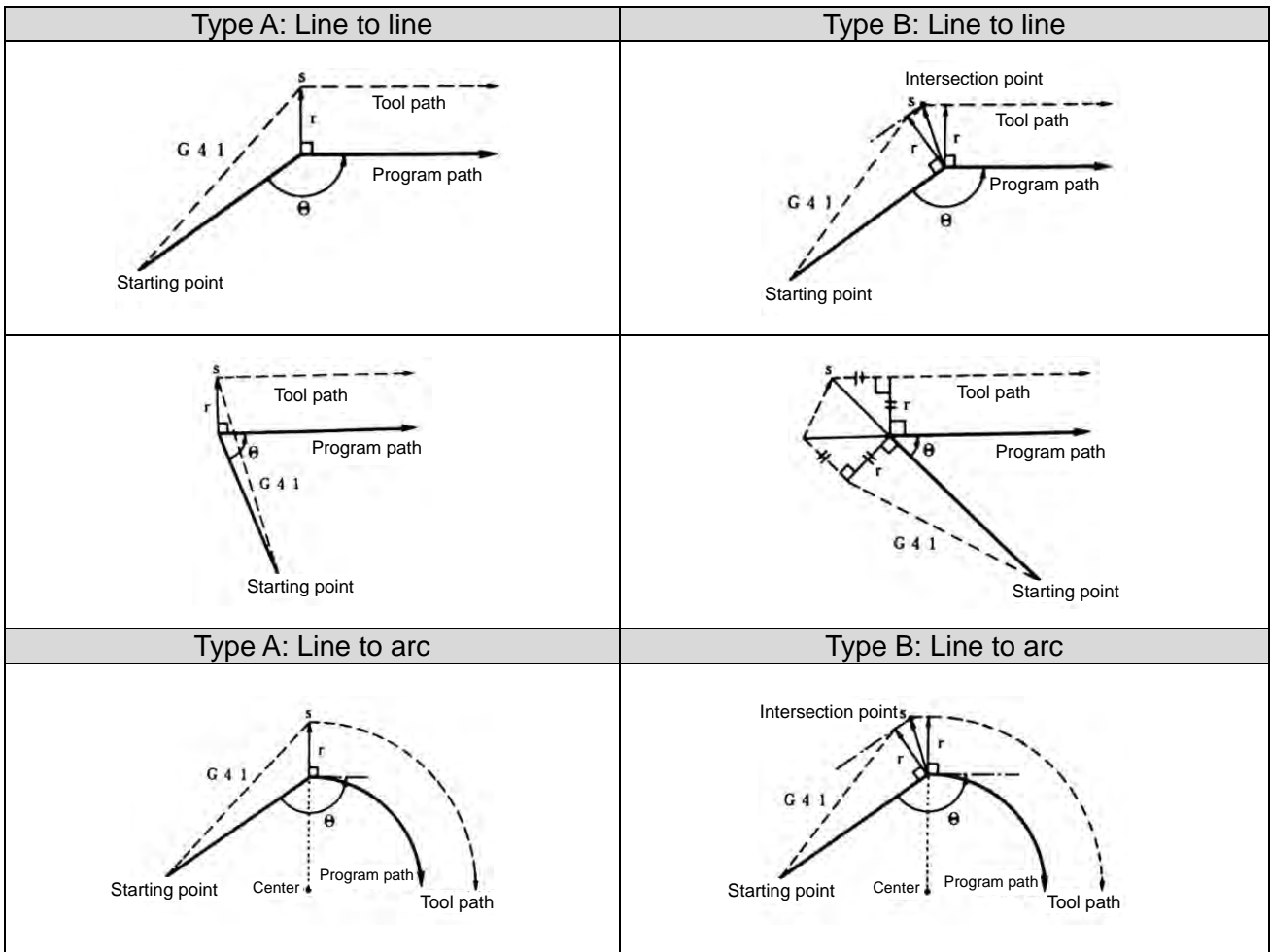
Inward curved angle: Arc to line	Outward curved angle: Arc to line

Inward curved angle: Arc to arc	Outward curved angle: Arc to arc

Compensation path toggle switch: When a motion path without compensation enters a path with compensation, the motion track of the tool center is as shown in the figure below. During the compensation period, the motion track remains active. When compensation path execution is cancelled (G40) or switched to compensation in another direction directly, the motion track will be shown as the figure below.

G40→G41	G40→G42
G41→G42	**G42→G41**
G41→G40	**G42→G40**

G43/G44: Tool Length Compensation Command

Format: G43 Z_ H_
         G44 Z_ H_

> G43: Tool length positive compensation. For positive tool length the tool axis moves in the positive direction.
> G44: Tool length negative compensation. For negative tool length the tool axis moves in the negative direction.

Description: This command enables operators to define individual tool length ID for length compensation. It ensures machining depth complies with program specifications and simplifies the program designation tasks.

> Z: The coordinate position is: zero point + tool length compensation. The zero point is the reference to the Z axis of tool coordinate.
> H: Tool length compensation data ID expressed in binary digits. The tool length compensation represented by given ID will be taken as the height compensation of this program. H0 means canceling the compensation.

> G43 Z_H_: If the value represented by the compensation ID is positive, then tool compensates upward, otherwise, it compensates downward.
> G44 Z_H_: If the value represented by the compensation ID is positive then tool compensates downward, otherwise, it compensates upward.

Note:
(1) The system cancels the tool length compensation value automatically before executing commands G53, G28, and G30 when tool length compensation is active. Later the program runs without tool length compensation unless another H_is assigned by command G43/G44.

(2) Parameter 307 is for setting the moving mode of tool length compensation when G43/G44 and G49 execute without specifying value Z. When the Parameter 307 is set to 0, it means G43/G44 and G49 move the tool length compensation without Z value specified. If Parameter 307 is set to 1, it means when Z value is not set in G43/G44 and G49, the compensation is completed by the internal system.

(3) The active tool length compensation (by G43 or G44) remains active when G28/G30 reaches the reference point and then cancelled when returned to the mechanical origin. The tool length compensation will not be resumed in later motion blocks.

(4) The active tool length compensation will be cancelled by the system and switched to G49 status when commands M30 and M02 are executed successfully.

(5) The active tool length compensation will be cancelled by the system and switched to status specified by G49 when a RESET signal is received by the system.

G49: Tool Length Compensation Cancelling Command

Format: G49

Description: The tool length compensation function is a status assignment. After compensation function is performed, G49 is required to cancel the tool length compensation function.

G50/G51: Scaling Up/Down Command/Cancelling Command

Format: G51   X_ Y_ Z_ P_

      X_ Y_ Z_: Coordinates center for scaling
      P_0: Scaling ratio

Description: X_ Y_ Z_ sets the X, Y, and Z coordinates of the scaling center and P sets up the scaling ratio. The machining path is enlarged or contracted with the coordinate's center given by X_ Y_ Z_ and scaling ratio by P. The minimum value of P is 1 with a ration range between 0.001 and 999.999. For example, P100 represents to scale down to one-tenth. P value can be set to negative to scale in mirror image. For example, G51 X0 Y0 P-1000 is the mirror image of path with origins of X0Y0.
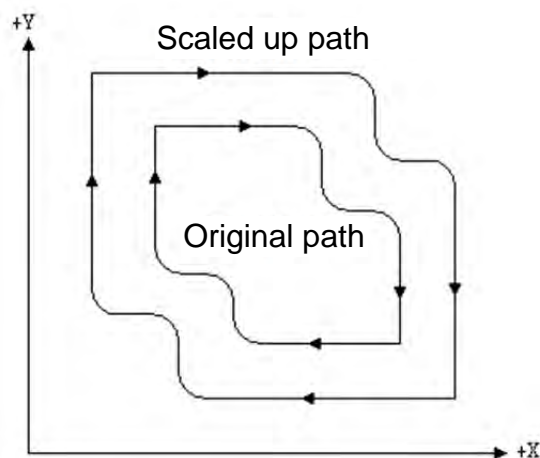
When executing commands M02 and M30 or when the NC200 controller is reset, the scaling mode will be cancelled. Pressing the reset key can also cancel the scaling. To return to the status with original scale and cut in normal path, execute command G50.

G52: Local Coordinate System Setup Command

Format: G52 X‐ Y‐

      X‐ Y‐: Origin of the local coordinates system

Description : Before setting command G52, please use one of the coordinate system of G54 ~ G59. An absolute value should be added after command G52 and a local-coordinate system can be defined. Command G52 specified as zero cancels the local coordinates system setup.

G53 : Mechanical Coordinate System Setup Command

Format: G53 X‐ Y‐ Z‐

      X‐ Y‐ Z‐: Actual arriving position of mechanical coordinates

Description: Coordinates X, Y, and Z are the actual ending point in mechanical coordinates system specified by the program coordinates. Machine suppliers use this command to set up the tool replacement position with the reference point given in mechanical coordinates. The command format must be specified as absolute; G53 with increment format will not be executed. Command G53 is a non-continuous G command and is valid for single block containing it. After the G53 command is executed, the system moves in G00 mode, the tool radius compensation is paused and tool length compensation cancelled automatically; the former resumes at the next motion block and the latter becomes active only by re-assigning it again.

Note:
    (1)   Command G53 functions only with status given by G90. It is ignored when under status specified by G91. However, G00/G01 or G90/G91 will change the system status and still effective in changing the motion status of the next block.
    (2)   If the block containing command G53 also contains a specific axial command, then the axis moves to a specified point. Otherwise, there is no position command.
    (3)   If both commands G53 and G28 are set in the same block, the one read later becomes active. When command G53 is active, the motion position refers to the mechanical coordinates. If command G28 is active, then the absolute coordinates are referred to.

G54 ~ G59 : Workpiece Coordinate System Selection Command

Format : G90 G54 X_ Y_ Z_ or ;
            G90 G55~ G59 X_ Y_ Z_

Description: Command G54 ~ G59 assigns any one of the six fundamental coordinates system as the workpiece coordinates system. A workpiece coordinates system is created by moving the tool from the mechanical origin to the desired program origin (with proper X and Y distance), registering this position data in the workpiece coordinates system setup (G54 ~ G59) in OFS group , and executing the workpiece coordinates system code, then you can set up the workpiece coordinates origin. The system also features a designation function out of 64 sets of expansive workpiece coordinates system options. This is done by assigning values to P_ (with valid value range of 1~64) in command G54. For example, G54 P10 X_ Y_ Z_. It means the tenth coordinates system of expanded workpiece coordinates system is used.

G61: Exact Stop Mode

Format: G61

Description: Command G61 functions the same as command G09 does, except that the latter is a non-continuous status command while the former is a continuous statement. After command G61 is executed, each execution of commands G01, G02, and G03 instruct the system to decelerate to fully stop for inspection. This mode remains active until it encounters command G64 (cutting mode).

_____ Tool path in G61 mode.

- - - - - Tool path in G64 mode.
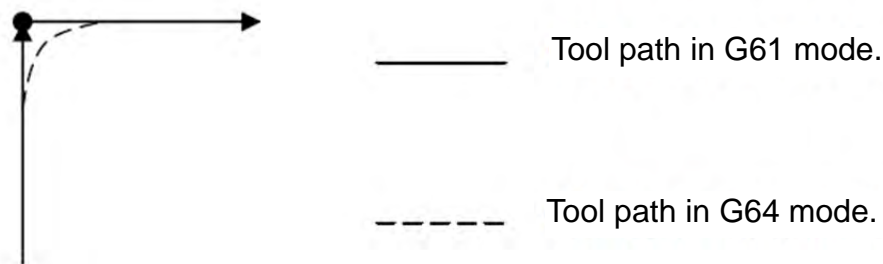
G64: Cutting Mode Command

Format: G64

Description: After command G64 is executed, the system remains moving at a certain speed to transit into the execution of the next motion block instead of decelerating to full stop at the end of each motion command. Normally, the initial status of the system is set to G64 cutting mode. When command G64 is used, the tool path of NC machines becomes smoother when machining. Command G64 differs from G61 in that it cuts at a constant feed rate and does not decelerate to a full stop between motion blocks.

G65: Non-continuous Effect Macro Calling Command

Format: G65 P_ L_I

       P_: Program number
       L_: Number of repetitions
       I_: Value of independent variable

Description: Command G65 calls a macro program. Macro program makes a subroutine with different types of control functions such as variables, calculation commands, and control commands. In the main program, a macro is active only when it is called. This command works the same way as M98 does except that it is non-continuous.

NC position	Local variable	NC position	Local variable	NC position	Local variable
A	#1	I	#9	T	#20
B	#2	J	#10	U	#21
C	#3	K	#11	V	#22
D	#4	M	#13	W	#23
E	#5	Q	#17	X	#24
F	#6	R	#18	Y	#25
H	#8	S	#19	Z	#26

Command G65/G66 can nest macros up to 8 layers. When used together with subroutine calling command M98, the program nest layers remain at eight.

G66/G67: Continuous Effect Macro Calling Command /Cancelling Command

Format: G66 P_ L_I
       Or G67

       P_: Program code
       L_: Number of repetitions

       I_ : Value of independent variable

Description: Command G66 functions the same as G65 except that instead of being active for a single block, the former keeps on calling the macro in later statements until it is canceled by command G67.

G68/G69: Coordinate System Rotation/Cancelling Command

Format: G68 X_Y_R_

> X_Y_: Coordinates of the rotation center
> R_: Rotation angle; positive value for counterclockwise rotating and negative for clockwise.
> The system rotates in units of 0.001 degree and in range from zero to 360 degrees.

Description: Command G68 can regard a given zero point as the center for coordinates rotation. With this command, users may rotate the coordinates of the machining path according to the specified rotation angle. G68 can be set with absolute command (G90) as well as incremental (G91).

G73: Peck Drilling Cycle Command

Format: G73 X_ Y_ Z_ R_ Q_ F_ K_ D_

> X_ Y_: Ending position of a single block
> Z_: Bottom of hole to be drilled
> R_: Initial safety height
> Q_: Depth of each peck drilling
> F_: Feed rate
> K_: Number of cutting loops
> D_: Retreat distance of every cutting

Description: This command sets the machine to retreat a constant distance "d" after drilling a distance (depth) of "Q" and keeps on drilling to the desired hole depth of "Z". Here value Q is an absolute value. Value d is the moving distance specified by parameter D. If the setting value is 1 mm, it means the default value of retraction amount d is 1 mm. See the figure below for the operation specified by this command.
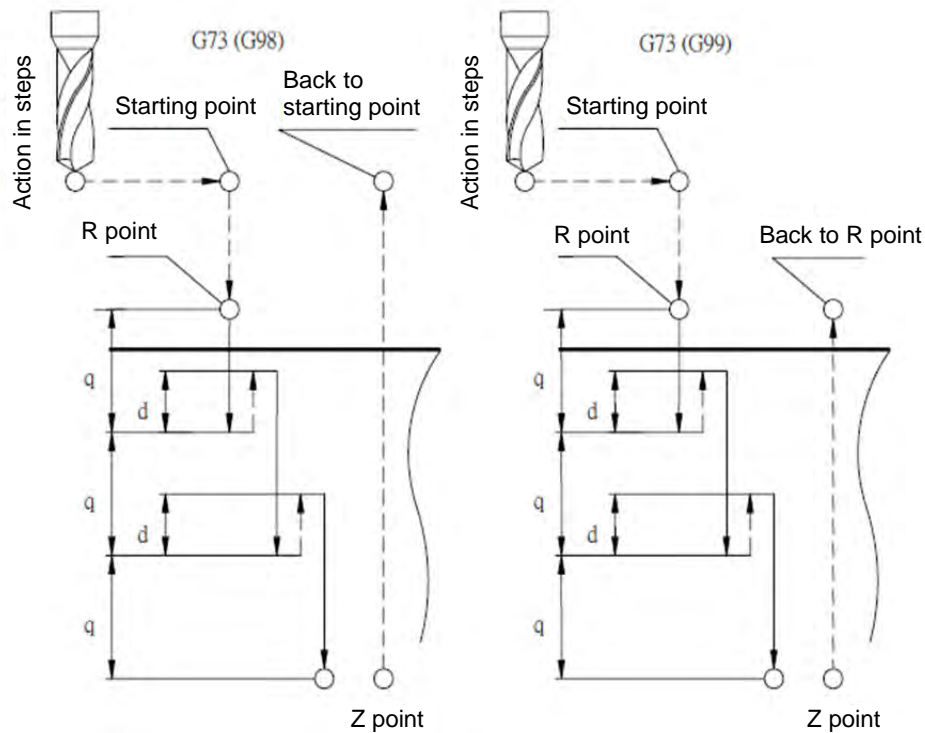
Note:

    (1)   When executing peck drilling loop command, the tool radius compensation function will be ignored.

    (2)   Execute command G80 to cancel the motion of loop cutting.

    (3)   Value Q cannot be specified as a negative value. If this setting value is negative, it will be regarded as positive.

    (4)   The value of parameter K will be rounded down to the nearest whole number; e.g., K2.6 round down to K2 and K0.6 to K0.

    (5)   When K value is in defined as absolute, it executes the cycle command for specified number of times at its original position. When the value is defined as increment, it executes the cycle command for specified number of times according to the specified distance.

G74 : Left Spiral Tapping Cycle Command

Format: G74   X_ Y_ R_ Z_ P_ F_ K_

X_ Y_: Ending point of a single block
Z_: Bottom of hole to be tapped
R_: Initial safety height
P_: Pause time (in the least unit of 1/1000 second), without decimal point.
F_: Spiral feed rate
K_: Number of loops

Feeding speed of tapping (mm/min) = Lead (mm/rev) × Spindle speed (rev/min).
F = P × S

Description : This command is for left-handed threading, which requires left-handed tapping cutter and reverse turning spindle.
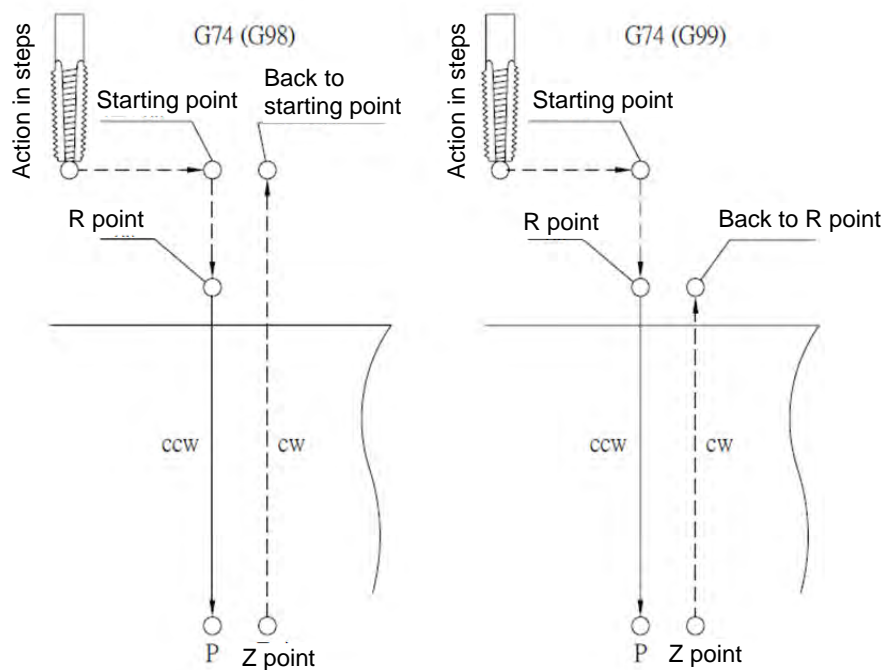
In tapping cycles, the machine cuts at the speed given by parameter F (100% program value); functions of spindle revolution factor and feed rate of the control panel are inactive.

G76: Fine Boring Cycle Command

Format: G76 X_ Y_ R_ P_ Z_ Q_ F_K_

        X_ Y_: Ending position of a single block
        R_: Initial safety height
        P_: Pause time (in the least unit of 1/1000 second), without decimal point.
        Z_: Bottom of hole to be tapped
        Q_: Offset distance
        F_: Feed rate
        K_: Number of loops

Description: This command is for fine boring. When cutting into the given depth, the spindle stops turning and be in position after pausing for the time specified by P_. Then, the tool center moves away from the workpiece surface for a distance set by program Q_, thus the tool makes no contact with the workpiece surface. In this way, the tool is able to return to height R or staring point without scratching the workpiece.
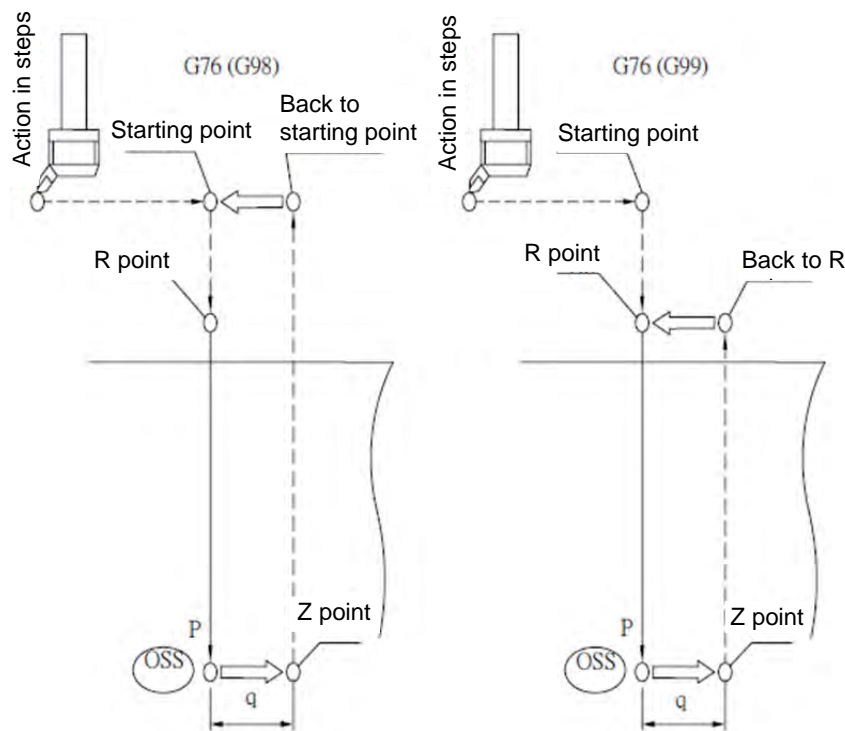
Figure 1: Fine boring cycle
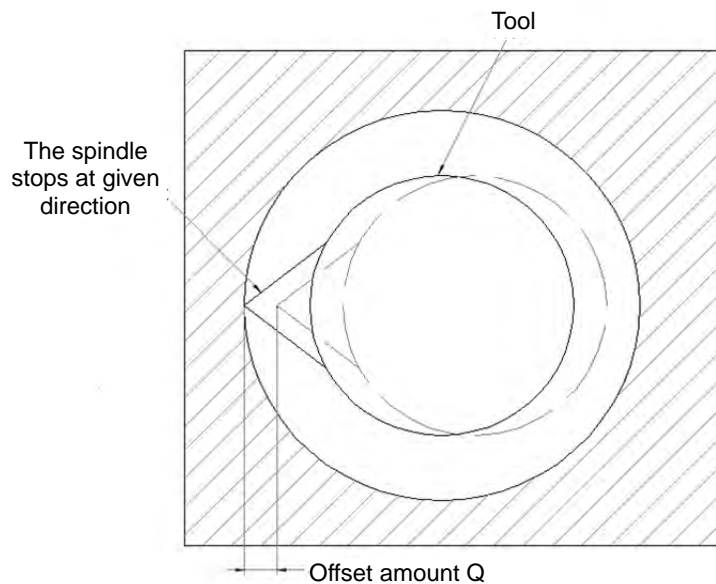
Figure 2: Offset amount for fine boring cycle

See below for movements set by this command (refer to Figure 1):
The firstly boring cutter fast positions to point (X, Y) and height R. Then, it cuts to depth Z with F feed rate before the spindle stops and positions. The cutter tip points to the positioning direction; offset the boring cutter for distance Q away from the hole wall (Figure 2). In this way, the cutter can move out of the hole without scratching the machining surface. Then, the tool center moves back to the original center position after the boring cutter moves to point R or the starting point. Finally, the spindle resumes turning.

The offset amount shown in Figure 2 is given by parameter Q. The value of parameter Q must be positive (if a negative value is given its absolute value will be used instead; to offset 1.0 mm, set the parameter to Q1.0). The offset direction can be set by coordinates (+X, +Y) or (-X, -Y). Parameter Q should not be set too big as the tool center may collide with the workpiece.

G80: Cycle Cancelling Command

Format: G80

Description: All cycle commands are status commands. Please cancel the cycle command status before resuming normal cutting operations. Command G80 cancels the cycle status set by commands G73, G74, G76 and G81 ~ G89.

G81: Drilling Cycle Command

Format: G81 X_ Y_ Z_ R_ F_K_

> X_ Y_: Ending position of a single block
> Z_: Bottom of hole to be cut
> R_: Initial safety height
> F_: Feed rate
> K_: Number of cycles

Description: Command G81 is used for drilling loops of general purpose. It drills to given depth Z in one shot without any retreat. See the figure below for reference.
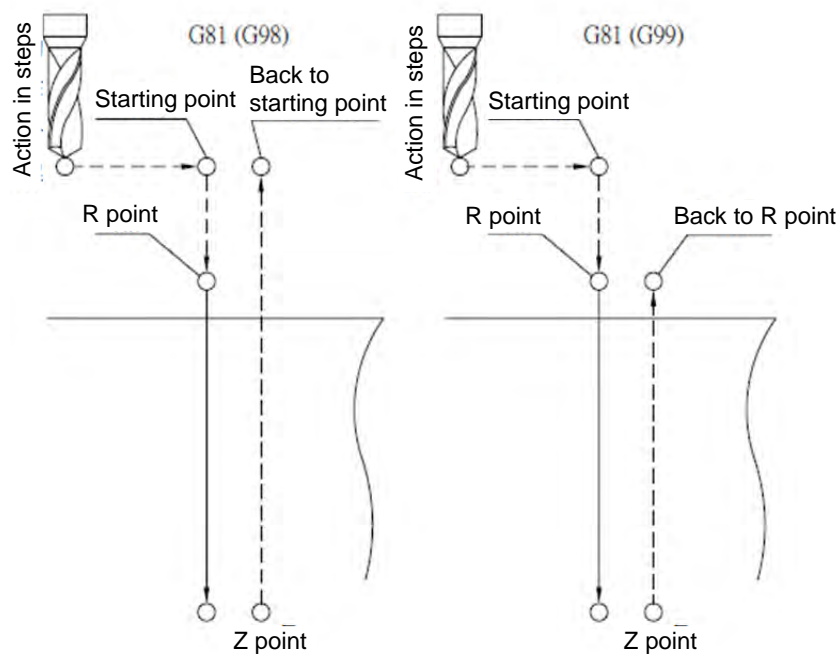
G82：Countersunk Drilling Cycle Command

Format: G82 X_ Y_ R_ Z_ P_ F_K_

        X_ Y_: Ending position of a single block
        Z_: Bottom of hole to be cut
        R_: Initial safety height
        P_: Pause time (in the least unit of 1/1000 second), without decimal point.
        F_: Feed rate
        K_: Number of cycles

Description: Command G82 functions the same as command G81 except that the former can designate a pause time P_ in a single block after it drills to the hole bottom.
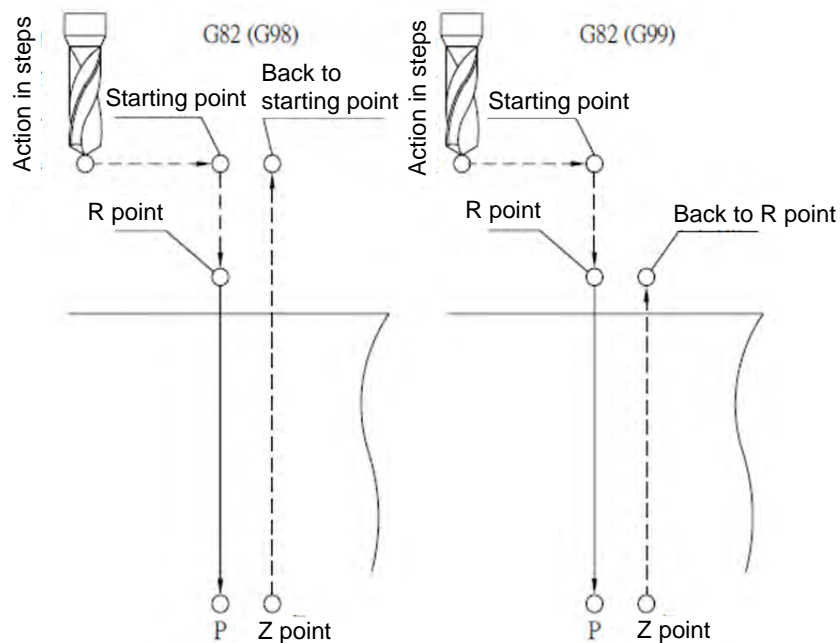
G83: Deep Hole Peck Drilling Cycle Command

Format: G83 X‿ Y‿ R‿ Q‿ Z‿ F‿ K‿ D‿

        X‿ Y‿: Ending position of a single block
        Z‿: Bottom of hole to be cut
        R‿: Initial safety height
        Q‿: Peck cutting depth
        F‿: Feed rate
        K‿: Number of cycles
        D‿: Lead d of every drilling

Description: Command G83 functions the same as command G73 except that the former returns back to height R after each deep drilling so that cutting debris can be fully removed and cutting fluid may flow into the hole for better cooling.
The machine is set to move as described below:
The tool returns back to point R after cutting to depth Q. Then, it is positioned to a point higher than the last drill ending point of distance "d". Next, it starts drilling for depth "q+d" in the next drilling process. And it drills to given position Z before returning back to position R or the initial point to finish drilling a deep hole.
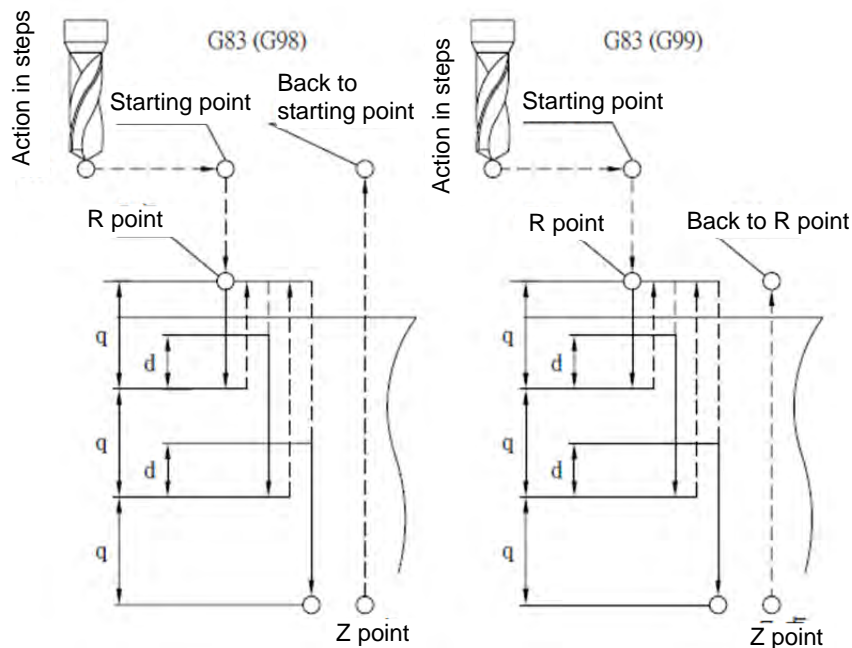
G84: Right Spiral Tapping Cycle Command

Format: G84   X_ Y_ R_ Q_ Z_ P_ F_ K_

           X_ Y_: Ending position of a single block
           Z_: Bottom of hole to be cut
           R_: Initial safety height
           P_: Pause time (in the least unit of 1/1000 second), without decimal point.
           F_: Spiral feed rate
           K_: Number of cycles

Description: Command G84 taps right-handed threads. It functions the same as command G74 except that the turning direction of G84 is opposite to that of command G74. All the remaining operations and notes are the same as command G74.
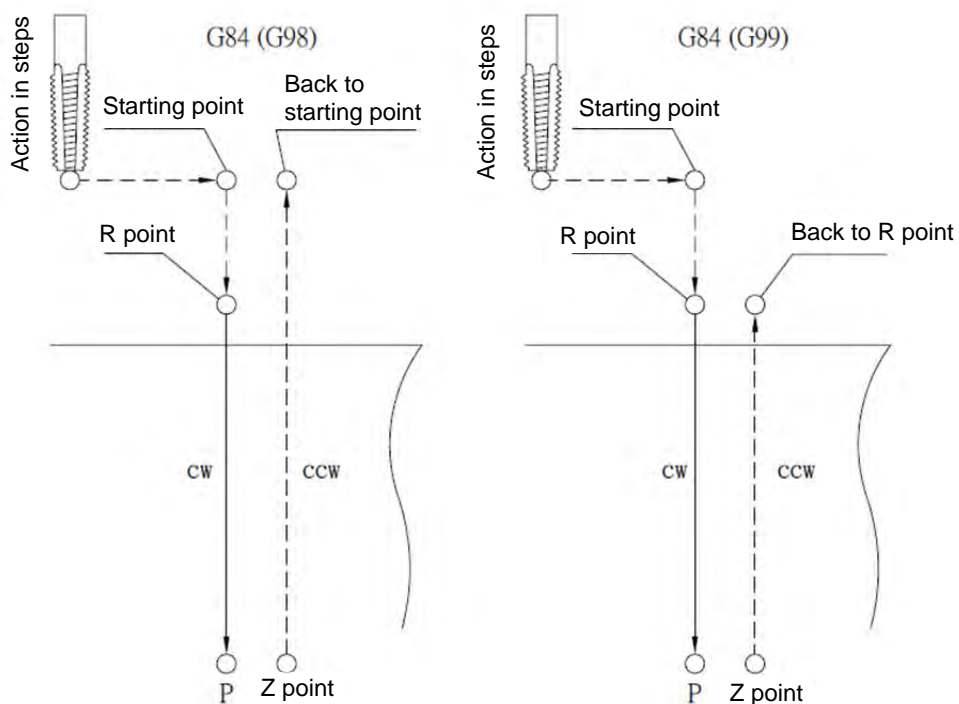
G85 : Broaching Cycle Command

Format: G85 X‾ Y‾ R‾ Z‾ F‾ K‾

X‾ Y‾: Ending position of a single block
Z‾: Bottom of hole to be cut
R‾: Initial safety height
F‾: Feed rate
K‾: Number of cycles

Description: Command G85 improves boring precision significantly. It usually works with reaming or boring cutter for holes with high reaming or boring accuracy.
See below for the movements set by this command:
The tool cuts to given depth Z from point R in given feed speed F before lifting the up to point R at the same feed speed of F. Then, it returns to point R (G99) or initial point (G98).
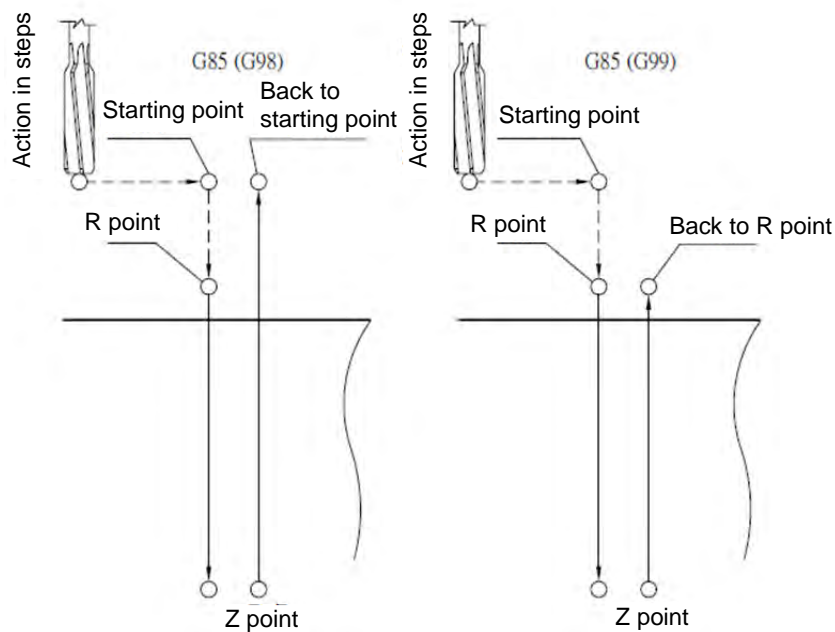
G86: Rough Boring Cycle Command

Format: G86 X‿ Y‿ R‿ Z‿ F‿ K‿

        X‿Y‿: Ending position of a single block
        Z‿: Bottom of hole to be cut
        R‿: Initial safety height
        F‿: Feed rate
        K‿: Number of cycles

Description: See the figure below for the motions set by this command. After bored to given depth Z, the spindle stops turning and lifts to the initial height with fast feeding command; a full boring cycle is completed. The way it works is similar to G81. As the tool keeps contacting the working surface constantly and is not turning when lifting off the hole, the hole wall may be left with minor scratches.
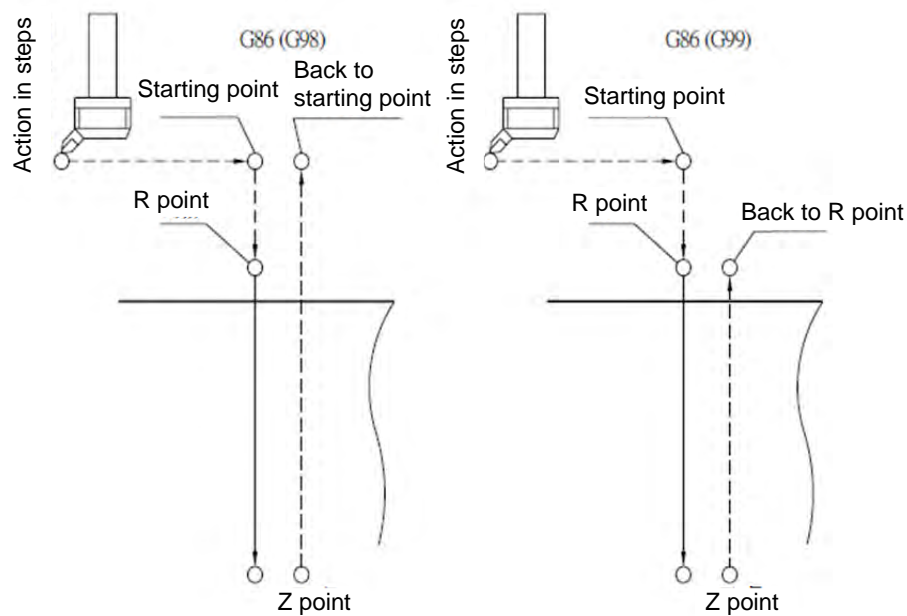
G87: Rear Boring Cycle Command

Format: G87 X‿ Y‿ R‿ Z‿ Q‿ P‿ F‿ K‿

        X‿Y‿: Ending position of a single block
        Z‿: Bottom of hole to be cut
        R‿: Initial safety height
        Q‿: Offset distance
        P‿: Pause time (in the least unit of 1/1000 second), without decimal point.
        F‿: Feed rate
        K‿: Number of cycles

Description: The tool fast positions to the position set by coordinates (X, Y) before the spindle moves to the position the cutter tip in a given direction. The tool center then offsets a distance of value Q. In this way, the cutter blade does not contact the hole wall when moving to height R. After moving to height R and complete fast positioning, the tool center moves to the original position given by coordinates (X, Y) and the spindle starts to turn forward. The tool now starts cutting from point R to point Z. The tool center then offsets a distance Q and the spindle positions after reaching point Z. The tool then returns back to the Z-axis initial point with fast positioning and resets the offset after reaching the initial point. The offset value Q equals the one set by command G76. Please note that command G87 does not support operation in G99 mode.
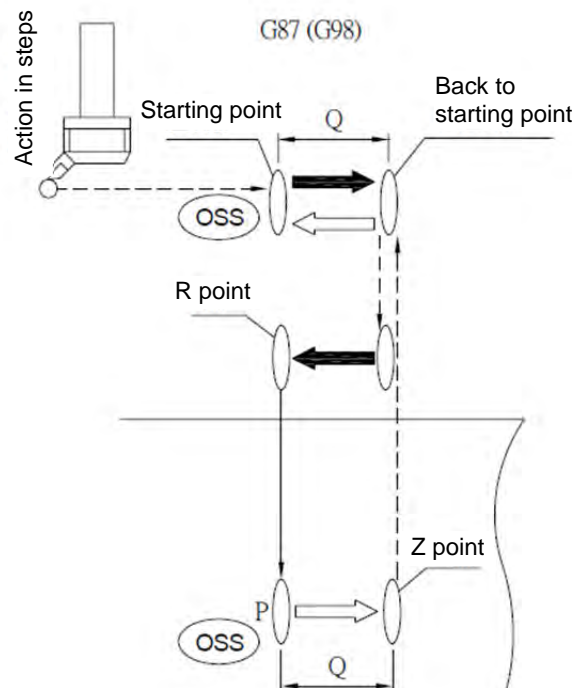
G88: Boring Cycle Command

Format: G88 X‿ Y‿ R‿ Z‿ P‿ F‿ K‿

X‿Y‿: Ending position of a single block
Z‿: Bottom of hole to be cut
R‿: Initial safety height
P‿: Pause time (in the least unit of 1/1000 second), without decimal point.

F‿: Feed rate
K‿: Number of cycles

Description: Command G88 sets the tool to cut from point R to given point Z and pause at point Z for the time specified in parameter P. The spindle stops turning and execution (same as executing command M00) after a pause period and fast retreats the tool to point R or the initial point only after the Cycle start key is pressed.
In addition, the operator may manually move (by switching the system to **MPG mode**) the tool upward in Z+ direction after cutting to point Z. Switch the system mode back to **Auto** and press the program execution key to resume program control. After the Z-axis is lifted to point R (G99) or the initial point (G98), a full cycle of command G88 is accomplished. This command is used for blind hole boring.
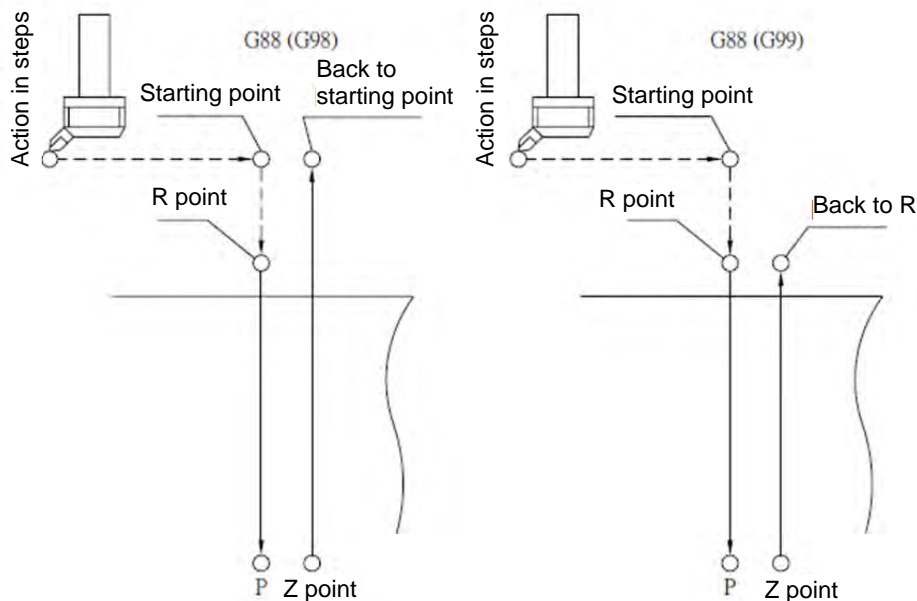
G89: Boring Cycle Command

Format: G89 X‿ Y‿ R‿ Z‿ P‿ F‿ K‿

X‿Y‿: Ending position of a single block
Z‿: Bottom of hole to be cut
R‿: Initial safety height
P‿: Pause time (in the least unit of 1/1000 second), without decimal point.

F‿: Feed rate
K‿: Number of cycles

Description: Command G89 is for blind hole reaming. It functions the same as command G85 except that a pause P can be added at point Z. With the added pause at point Z, the machining accuracy for depth and diameter of the hole can be greatly improved by keeping the tool staying for a while when cut to position Z.
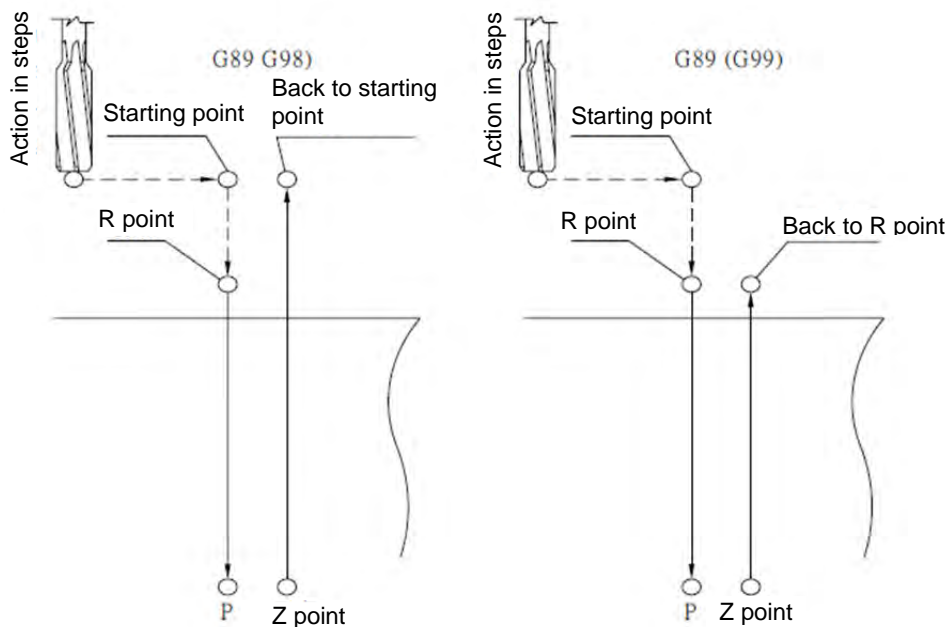
G90: Absolute Coordinate System Command

Format: G90 X‿ Y‿ Z‿

Description: Command G90 is a status command of continuous effects. When this command is executed, all axial commands and coordinate works with their absolute values specified. That is, the tool moves relative to a reference point of the origin in the workpiece coordinate system. Each axial command after command G90 moves an actual distance relative to the origin of the workpiece coordinates.

G91: Incremental Coordinate System Command

Format: G91 X̱ Y̱ Ẕ

Description: The increment command G91 sets all axial movements in a motion program of a single block. It will move or rotate incrementally from the current position to the specified position. The G91 command is a status command that cancels command G90 once it is activated.

G92 : Coordinate System Setup Command

Format: G92 X̱ Y̱ Ẕ

Description: Command G92 X0 Y0 Z0 sets the current tool position as the zero point of an absolute coordinates system. The absolute command in the program calculates positions relative to this point. Values of absolute coordinates and current positions are all refreshed with new values given by parameters of command G92 X̱ Y̱ Ẕ (as long as any of the three parameters X, Y and Z have values assigned to them).

Note:
(1)   When G92 is active, the program will keep running until encountering commands for end of program (M02 / M30) and then the status set by G92 will be canceled.
(2)   To reset the status set by command G92, press the RESET key.

G94: Feed Rate (mm/min) Setup Command

Format: G94 G01 X̱ Y̱ Ẕ F̱

Description: The feed rate set by command G94 is in the unit of mm/min. It sets the tool to cut at the speed given by parameter F. It is commonly used by most milling machines for feed rate calculation.
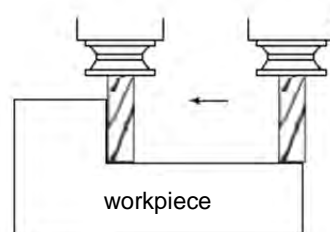
Feed rate in unit of
maintenance/min or inch/min

G98: Return to the Initial Point of the Fixed Cycle

Format: G98 G8_ X_ Y_ Z_ R_ F_

Description: Command G98/G99 is a status command for returning the tool back to a specified height after the execution of a fixed loop command ends. Command G98 returns the tool back to its initial point after a cycle command. Command G98 can be replaced by command G99.
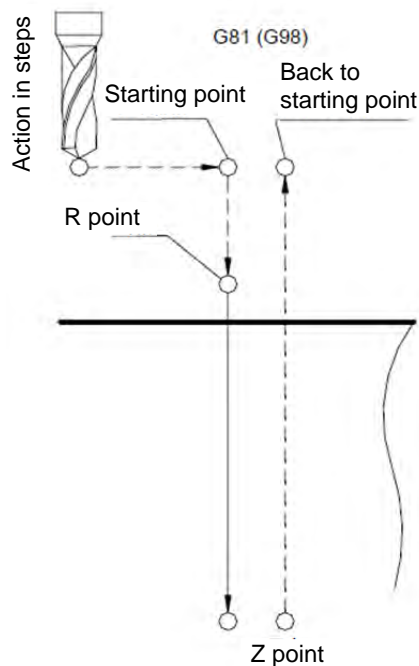
G99: Return to the R Point of the Fixed Cycle

Format: G99 G8_ X_ Y_ Z_ R_ F_

Description: Command G99 returns the tool back to its reference point (R) after a cycle command. Please use command G98, instead of G99, to return the tool back to its initial point after a cycle command.

Chapter 13   M Commands

The auxiliary function M commands are used in turning the on and off functions of the machine. Its format is a capital letter M suffixed with three digits. Some systems-defined M commands can be executed for program control without any MLC coding.

13.1 M00 Program stop (non-optional)

Description: When an M00 command is executed when program running, the program stops immediately after the line containing the M00 command. This means that, the program stops at where the M00 command is executed. To resume the program execution, press the program execution key once again. The M00 command is used for inspecting tools or workpiece's appearance and dimensions when cutting.

13.2   M01 Program stop (optional)

Format：M01；

Description: M01 command functions the same way as M00 command does except that it does not stop a program from running on its own. It comes effect only when the optional stop key on the secondary control panel is pressed. If the optional stop is not enabled, the program ignores M01 command and keeps on running until executing an M00 command or program ending command with optional stop function enabled.

13.3   M02 End of program

Format: M02；

Description: M02 command is commonly placed at the end of a machining program to instruct the controller that the running program is to be ended. For an M02 command within a program, the content that follows M02 will be ignored and is regarded as program finished; the cursor stops at the block of M02 command.

13.4   M30 End of program, return to start

Format: M30；

Description: M30 command is commonly placed at the end of a program to instruct the controller that the running program is to be ended. When an M30 command is located within a program, the program stops running after executing M30 and then returns the cursor back to the beginning of the program.
M30 and M02 commands function the same way except that M02 command keeps the cursor at the block of M0 command while M30 command returns the cursor back to the beginning of the program.

13.5   M98 Subroutine call

Format: M98 P̲ L̲；

Description: You may group fixed or repetitive actions in a program into a subroutine to simplify the structure and reduce the length of the program. The main program calls a subroutine, which in turn can call another subroutine in next level, up to eight layers. When the controller reads an M98 command, it jumps to the designated subroutine and executes it according to the setting number of times.

P__: indicates program code of the subroutine; L__: indicates the number of times the subroutine is to be executed.

[Example]

Main program	Subroutine
N0010；	O1010；
N0020 M98 P1010；	N1020；
N0030；	N1030；
N0040；	N1040；
N0050；	N1050；
N0060；	N1060；
N0070；	N1070；
N0080；	N1080；
N0090；	N1090；
N0100；	N1100 M99；

13.6   M99 Return from subroutine

Format: M99；

Description: M98 command can run a subroutine from the main program. If the user desires
to return to the main program, issue M99 command again. Thus, after
executing to the block of M99, the subroutine will call the next block and then
start to execute the command. If P_ is added as the suffix of M99, the
subroutine will not return to the next block of M98 P_L_, but the P program
series of the main program.

[Example]

Main program	Subroutine
N0010；	O1010；
N0020 M98 P1010；	N1020；
N0030；	N1030；
N0040；	N1040；
N0050；	N1050；
N0060；	N1060；
N0070；	N1070；
N0080；	N1080；
N0090；	N1090；
N0100；	N1100 M99 P0060；
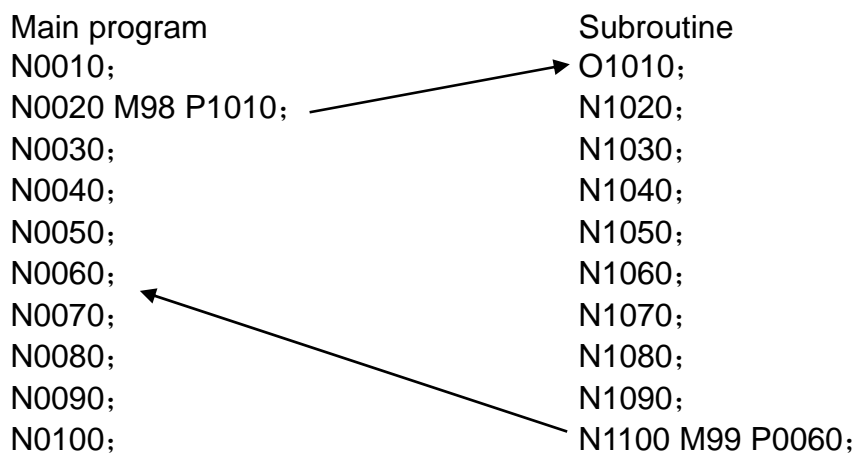
13.7   Table of commonly used M codes

The commonly used **M** codes are summarized in the table below. The functions of an **M** code are defined in the MLC except those pre-defined by the system. Actual functions of the **M** codes vary with individual machines.

M code	Function	Remarks
M00	Program stop (non-optional)	System defined
M01	Program stop (optional)	System defined
M02	End of program	System defined
M03	Spindle on clockwise	
M04	Spindle on counterclockwise	
M05	Spindle stop	
M06	Tool change	
M08	Coolant On	
M09	Coolant Off	
M19	Spindle positioning	
M20	Release spindle positioning	
M28	Cancel rigid tapping	
M29	Rigid Tapping	
M30	Ends of program, return to start	System defined
M98	Subprogram call	System defined
M99	Return from subroutine	System defined

(This page is intentionally left blank.)

Chapter 14 MACRO & Variables

14.1 Variables

You can assign values of variables contained in an NC program. You can calculate and modify multiple variable values in advance to adapt a program for a range of conditions. A variable is formatted as a symbol "#" suffixed with a variable number.

Types of variables

Type	No.	Function	Read	Write
Local	#1 ~ #50	Local variables are used by sub-routines or macros for data registry or calculation. They can be referred by arguments. The calling sub-routine can nest up to 8 levels with calling arguments at each level referred to some local variables.	★	★
Global	#51 ~ #250	It is for data registry or calculation in sub-routine or macros.	★	★
Maintain	#1601 ~ #1800	Systems variables are used for reading and writing internal data of the system during an NC operation. This is the non-volatile maintain variable.	★	★
MLC bit output	#1801 ~ #1832	Read MLC message status (MLC > NC) with variable number (Variable #1801~#1832 for bit and #1833~#1848 for word).	★	
MLC word output	#1833 ~ #1848		★	
MLC bit output	#1864 ~ #1895	Write MLC message status (NC > MLC) with variable number (Variable #1864~#1895 for bit and #1896~#1911 for word).		★
MLC word output	#1896 ~ #1911			★

14.2 Variable syntax

Numeric values required by a program can be assigned by using variables. This empowers the program for better flexibility and universality as you can use the variables for mathematical operations.

(1) You can specify the scope of local variables:
      # i: the ith variable   (for 1<= i <=50)

(2) You can define the variable number with expressions:
      For variable # [A], the value of A must be in range between A and the upper limit of the system variable number, that is 1 <= A <= the upper limit of the systems variable number. Value of A shall not be smaller than 0 or be a negative number.

#[<expression>]	Description
# [#20]	(valid)
#[#20Δ3]	(valid) where Δ represents operators =, +, -, *, and /
##20	(Invalid), there cannot be two variable symbols (#) in a sequence.
#[#20] = …	(valid)
#20 = …	(valid)
#[#20 - #10] = …	(valid), no operator can precede an equal (=) symbol
#[- #20]= …	(valid)

14.3   Calculation commands

You can subject variables to a variety of calculations and use the result as a value for another variable, as a combination of several variables or as an alternative to other variables.

#i, #j, and #k can be replaced by constants.

Command	Symbol	Usage	Definition
Elementary arithmetic	$+$	#i = #j + #k	Addition
	$-$	#i = #j - #k	Subtraction
	$*$	#i = #j * #k	Multiplication
	/	#i = #j / #k	Division
	$=$	#i = #j	Substitution
	[ ]	#i = #j * [#p + #q]	Parenthesis
Function	SIN	#i = SIN [#k]	Sine
	ASIN	#i = ASIN [#k]	Arcsine
	COS	#i = COS [#k]	Cosine
	ACOS	#i = ACOS [#k]	Arccosine
	TAN	#i = TAN [#k]	Tangent
	ATAN	#i = ATAN [#k]	Arctangent
	ATAN2	#i = ATAN2 [#m, #n]	Arctangent; the angle is adjacent side #m and diagonal side #n
	ABS	#i = ABS [#k]	Absolute value
	FIX	#i = FIX [#k]	Round down
	FUP	#i = FUP [#k]	Round up
	ROUND	#i = ROUND[#k]	Round off
	SQRT	#i = SQRT [#k]	Squared value
	POW	#i = POW [#m, #n]	#m to the power of #n
	BIT	#i = BIT [#m, #n]	The value of the #n bit of a binary #m
Logic operator	&	#i = #j & #k	AND logic
	|	#i = #j | #k	OR logic
	∧	#l = #j^#k	XOR logic
	!	#i = ! #j	NOT logic
Constants	PI	PI = π	Pi
	TRUE	TRUE = 1	Return value 1 when the IF statement is true
	FALSE	FALSE=0	Return value 0 when the IF statement is true

14.4   Control flow

Change the program flow with WHILE~ENDW and IF~GOTO commands.

14.4.1   Repetitive execution

When the WHILE expression is true: The program loops from the statement under WHILE to the statement above ENDW. Otherwise, the program jumps to the statement under ENDW for execution.

WHILE
⁀
ENDW

Example:
WHILE[#80<=360.] (Enters the cycle for repetitive execution when #80 is less than or equal
                to 360.)
     WHILE[#60>=20.] (Enters the second cycle for execution when #60 is greater than or
                    equal to 20.)
          #60=#60-2.
     ENDW   (The second cycle ends when ENDW is encountered.)
          #80=#80+15.
          #50=#50-0.05

ENDW   (The first cycle ends when the second ENDW is encountered.)

14.4.2 Branch conditions

When the IF statement is true, the program branches (GOTO) to statement number N for execution. Otherwise, the program executes the next node again. When jumping to N, adding symbol [] is not required.

GOTON (It jumps to N un-conditionally when being used independently.)

Types of conditional expressions:

Conditional expressions	Description	Examples		
#j > #k	#j greater than #k	#i = #j > #k	TRUE FALSE	returns value: #i=1 returns value: #i=0
#j < #k	#j less then #k	#i = #j < #k	TRUE FALSE	returns value: #i=1 returns value: #i=0
#j == #k	#j equal to #k	#i = #j == #k	TRUE FALSE	returns value: #i=1 returns value: #i=0
#j >= #k	#j greater than or equal to #k	#i = #j >= #k	TRUE FALSE	returns value: #i=1 returns value: #i=0
#j <= #k	#j less than or equal to #k	#i = #j <= #k	TRUE FALSE	returns value: #i=1 returns value: #i=0
#j != #k	#j not equal to #k	#i = #j != #k	TRUE FALSE	returns value: #i=1 returns value: #i=0

Example:

```
#100 = 1.234;              (Assign value 1.234 to #100)
#100 = #101;              (Assign value of #100 to #101)
#100 = [[#101+#102]/2.0]; (Assign value of #100 to be the average of #101 and #102)
#100 = #102+2.;           (Assign value of #100 to be the sum of #102 and 2)
#100 = SIN[#102];

X-#100                    (Assign the negative value of #100 to X coordinate)
G1X#100Y#101;             (Assign the value of #100 to X coordinate and #101 to Y
                           coordinate)
G1X[#100];                (Assign the value of #100 to X coordinate)
G1X[#100+#101];           (Assign the sum of values of #100 and #101 to X coordinate)
G2X[#100*SIN[#102]];      (Assign the multiplication of value of #100 by SIN#102 to X
                           coordinate)
G1Z#100F#102S#103;        (Assign the value of #100 to Z coordinate, #102 to F, and
                           #103 to S)
```

(This page is intentionally left blank.)

Chapter 15 Wiring

This chapter explains the wiring of the CNC controller and the meaning of various signals along with the standard wiring diagrams for each mode.

15.1 Wiring of system interface and main power circuit

15.1.1 System interface wiring diagram
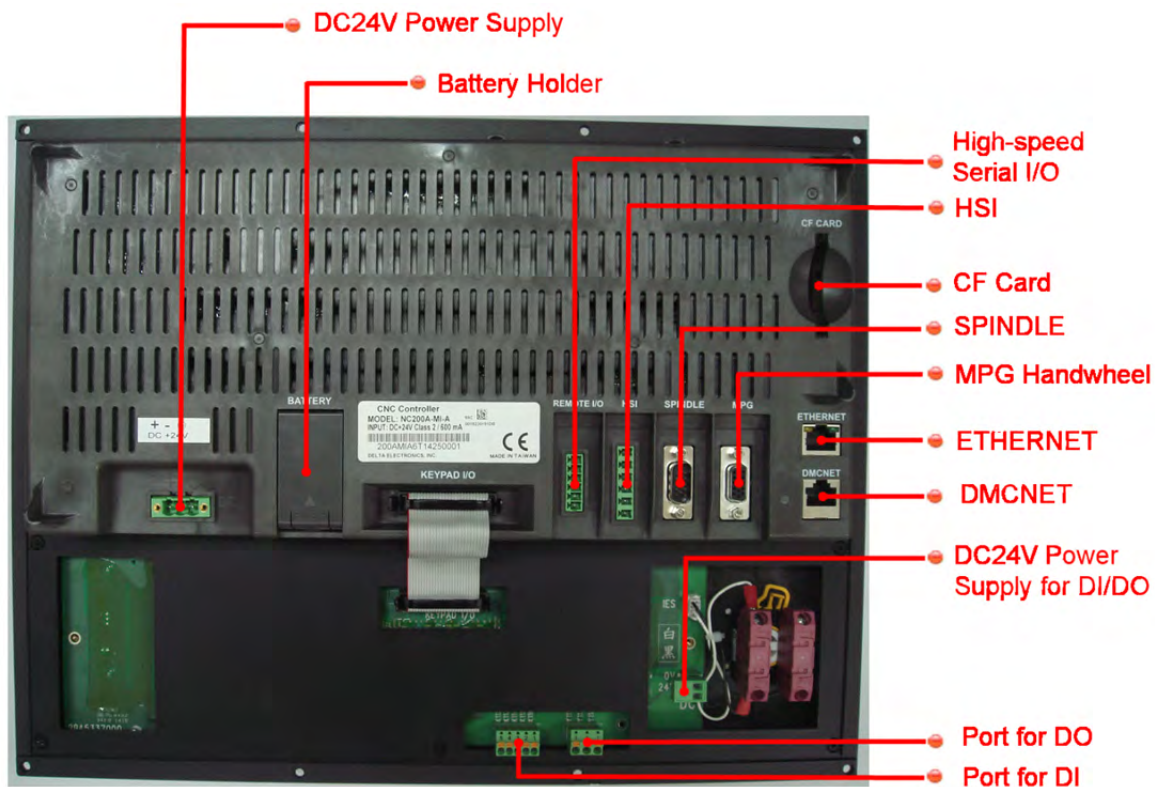
Connector	Description
High speed serial I/O	Every module features 32 pairs of inputs and outputs
	Up to 20 M between stations and up to 160 M (20 x 8) total length
MPG hand wheel	External hand wheel function with built-in 5 VDC power, 7 input points and 1 output point
Spindle	One input point for spindle encoder signal
	One output point for analog signal
	RS-485 connector
HSI	Two high speed counters One Emergency stop input

Connector	Description
Battery holder	24 VDC power input (Backup battery 3V)
DMCNET	High speed communication network connector
Ethernet network	DNC control and system monitoring
CF card	Storage for G code program
Emergency stop	Triggers system emergency stop when opening circuit
DC24V	DC24V input (for system and DI/DO power supply)
DI / DO	5 Digital input points
	3 Digital output points

Notes:

(1) Ensure correct DC 24 V power connection. Do not connect to AC 110 V power.
(2) On board and remote I/O require extra DC 24 V 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) power with the Y output.

15.1.2 Connectors and terminals of NC200 controller

Terminal mark	Name	Description		
0V、+24V ⏚	Controller power input	Connect the DC 24V power (15 W at 0.6 A)		
		Terminal ID	Wire color	Description
		+24V	Red	Positive of DC 24 V power
		0V	White	Negative of DC 24 V power
		⏚	Green	Power grounding
SPINDLE	Spindle specific connector	PIN ID	Description	
		P1	GND	
		P2	RS485 D+	
		P3	RS485 D-	
		P4	DAC_OUT	
		P5	DAC_GND	
		P6	NA	
		P7	+24 V	
		P8	SP_A+	
		P9	SP_A-	
		P10	+5V_OUT	
		P11	SP_B+	
		P12	SP_B-	
		P13	SP_Z+	
		P14	SP_Z-	
		P15	+5V_GND	
MPG	Hand wheel pulse input	7 DI input points and along with one hand wheel input terminal, A, /A, B, /B		
		PIN ID	Description	
		P1	External 24V input	
		P2 ~ P8	Digital Input	
		P9	Digital Output	
		P10	+5 V power positive end + (< 300 mA)	
		P11	XA+	
		P12	XA-	
		P13	XB+	
		P14	XB-	
		P15	+5V_GND	
REMOTE I/O	Remote I/O serial connection terminal	Remote I/O communication terminal, which ranges from X256 ~ X511 and Y256 ~ Y511. Every remote input and output is X32/Y32 and may connect up to 8 modules.		

Terminal mark	Name	Description	
		PIN ID	Description
		P1	TX+
		P2	TX-
		P3	RX-
		P4	RX+
		P5	GND
		P6	SHIELD
ETHERNET	Ethernet connector	Connect PC with RJ45 connector and regular type network cable. See below for wiring of the two ends.	
		PIN ID/color at end A	PIN ID/color at end B
		1. Orange & white	1. Orange & white
		2. Orange	2. Orange
		3. Green & white	3. Green & white
		4. Blue	4. Blue
		5. Blue & white	5. Blue & white
		6. Green	6. Green
		7. Brown & white	7. Brown & white
		8. Brown	8. Brown
DMCNET	DMCNET communication connector	Connect to Delta's servo drive ASDA-A2 -N with standard RJ45 connector. Wire in standard Ethernet manner.	
EMG	Emergency stop switch	Press to open the circuit.	
DI	External Input point	5 input ports are provided.	
DO	External Output point	3 output ports are provided.	
IES	EMG emergency stop contact	Emergency stop contact; EMG is enabled when open circuit. (It is wired as normal close)	

Notes:

(1) The IES connector is for the switch of EMG emergency stop. Activate EMG when breaking circuit.

15.1.3   Wiring the power line

The NC200 series controller is powered by direct power supply.

■   Wiring for DC power supply (to power on the controller)
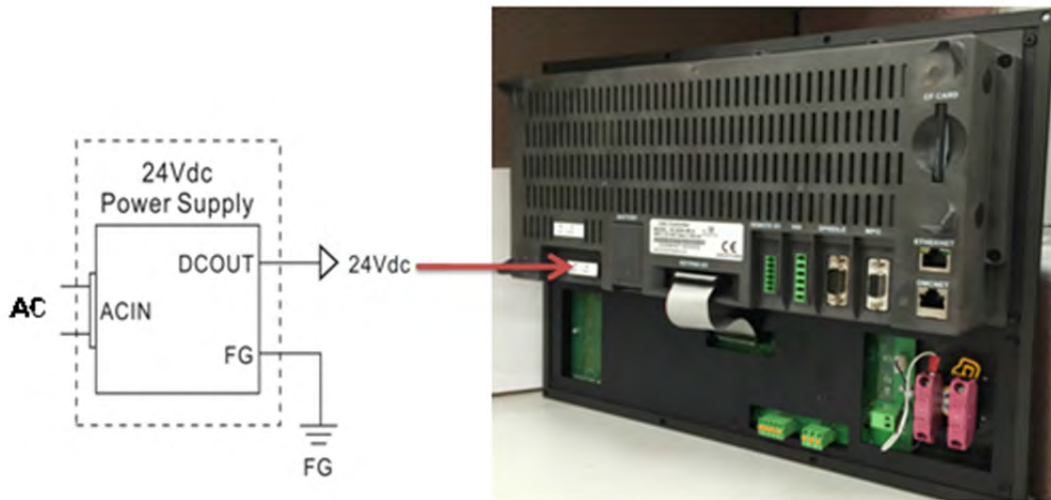
15.1.4   DMC connection setup for NC200 and ASDA-A2-N

Simple DMC Connection
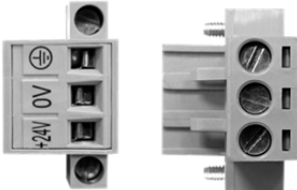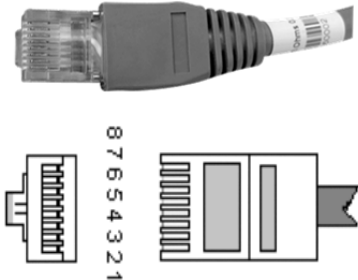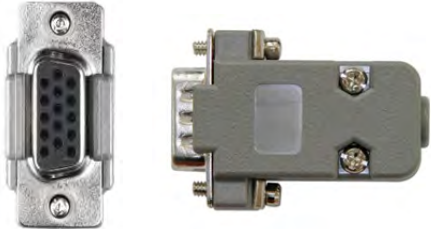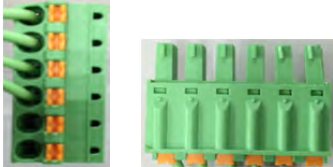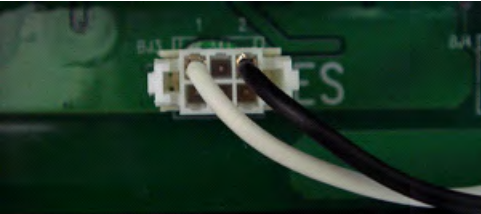
ASDA-A2-N and DMC connection setup
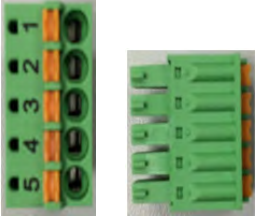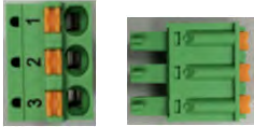
Set P1-01 to b
Set P1-01 (CW, CCW) to 010b
Set P3-00 (station ID) to 1~12
Set P3-01 (CNC connection) to 0203
Set P3-10 (communication) to 1
Set P2-15 (limit to the right) to 1XX
Set P2-16 (limit to the left) to 1XX
Set P2-17 (EMG) to 1XX

15.1.5   Connector Specifications of each NC200 port

Terminal name and mark	Connector	PINs
DC 24V power input		3
REMOTE I/O		6
DMCNET and ETHERNET (with standard RJ45 connector)		8
SPINDLE		15
MPG		15
HSI		6
IES		2

Terminal name and mark	Connector	PINs
DI		5
DO		3

Please use AWG30, UL1007 compliant I/O wires shielded with metal mesh to prevent signal noise. The DMCNET terminal uses RJ45 connector and insulated network cable to connect with the server.

15.2   I/O Wiring

15.2.1   Layout of DI 1 and DO 1 terminal blocks

The controller features external I/O with 5 configurable DI and 3 configurable DO for more flexible configuration. Pin definition is as follows.
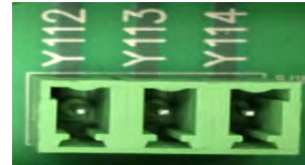
DI			
PIN 1	X112	PIN 2	X113
PIN 3	X114	PIN 4	X115
PIN 5	X116		

DO			
PIN 1	Y112	PIN 2	Y113
PIN 3	Y114		

15.2.2   Wiring of DI1 and DO1

The signal is input by voltage. (Permissible current voltage: **17 VDC ~ 32 VDC**; Surge current: under 50 mA)
Below is the wiring of DI. The servo drive applies external power supply.
NPN transistor (SINK mode)

NPN transistor (SINK mode)
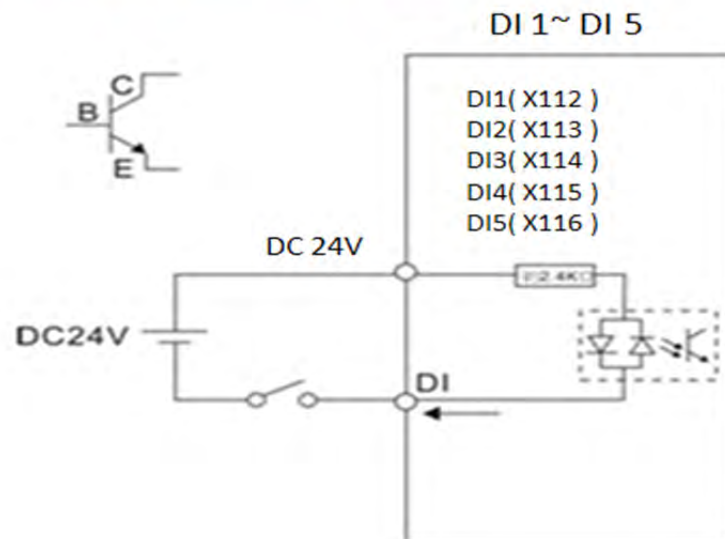
The drive connects to general load. (Permissible current: under 140 mA; surge current: under 200 mA)
Wiring of DO signal. The servo drive applies to the external power and the resistor is general load.

DO 1
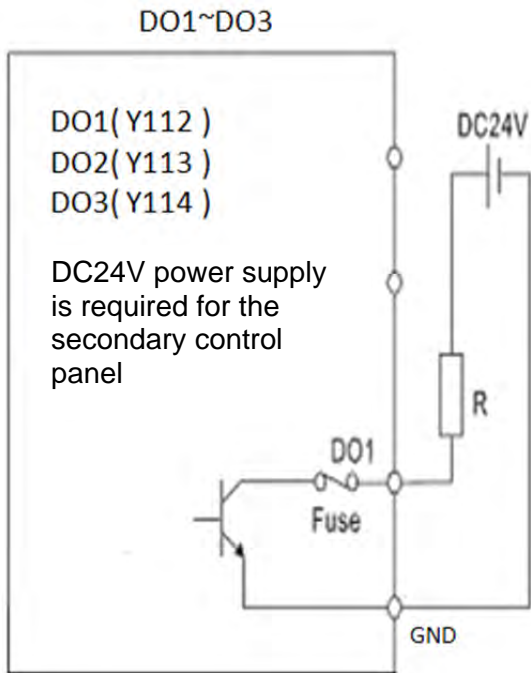
When inductive load is applied, using diode is suggested to protect the DO contact.
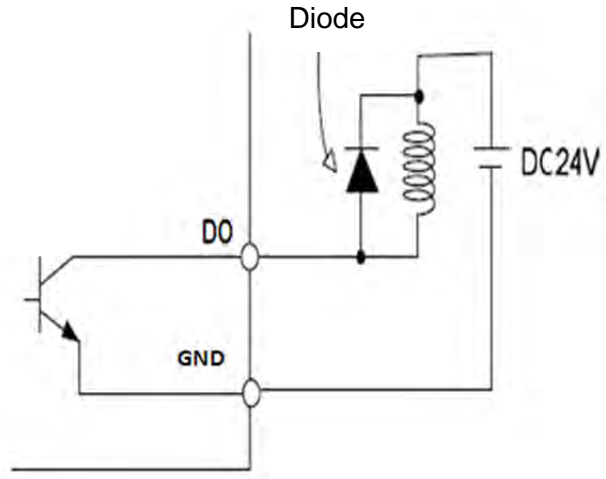
15.3   Wiring for SPINDLE terminal

15.3.1   Terminal layout of SPINDLE

NC200 series controller features one set of spindle feedback input and one set of spindle analog output.
See the figure below for its wiring:
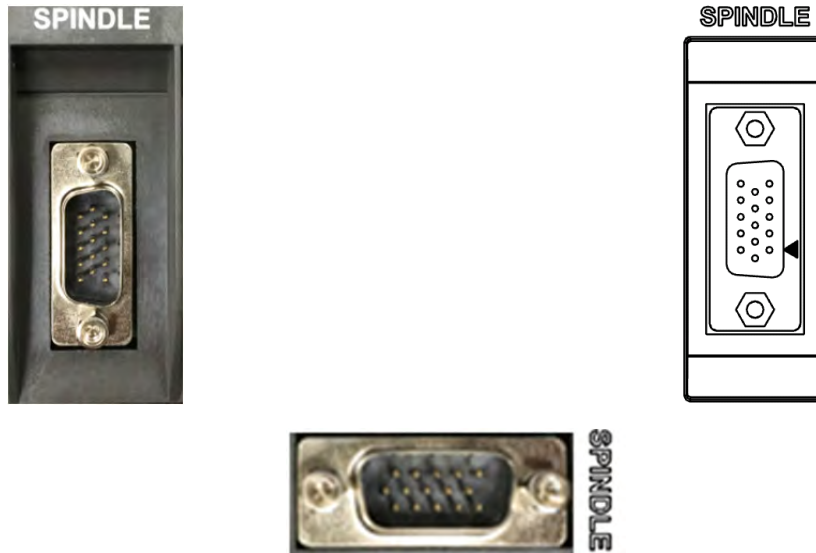
SPINDLE		
Pin No	Description	
PIN 1	GND	
PIN 2	RS-485_D+	RS485_D+
PIN 3	RS-485_D-	RS485_D-
PIN 4	SP_OUT	Spindle analog output
PIN 5	SP_GND	Spindle analog ground
PIN 6	NA	Reserved
PIN 7	+24V	+24V
PIN 8	SP_A+	Spindle encoder A phase positive input
PIN 9	SP_A-	Spindle encoder A phase negative input
PIN 10	+5V_OUT	Spindle encoder power output
PIN 11	SP_B+	Spindle encoder B phase positive input
PIN 12	SP_B-	Spindle encoder B phase negative input
PIN 13	SP_Z+	Spindle encoder Z phase positive input
PIN 14	SP_Z-	Spindle encoder Z phase negative input
PIN 15	+5V_GND	Spindle encoder power ground

15.3.2 RS-485 Connection

15 pin D-SUB male (RS-485) at NC200 wiring end	RS-485 at the wiring end of other device	Connector for controller
RS485_D+ (2)———————— RS485_D+ RS485_D- (3)———————— RS485_D-		 Top view

15.3.3 Wiring for analog spindle

Parameter Setting

399 Servo spindle: set to 1101
Analog spindle: set to 1020

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: rmp / 1: PPM	0~1
5	Bit 5: feedback form 0: high resolution (x1000) / 1: normal resolution (x4)	0~1

The SP1 in the channel setting has to be enabled and set to 10. 0 ~ 10 VDC corresponds to S0 ~S (maximum speed). The resolution of -10 V~+10 V is 14-bit.
Spindle analog output for Pin 4 and Pin 5:
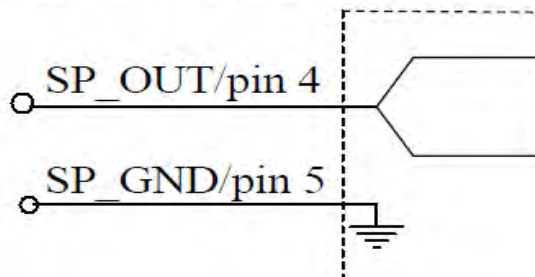
15.4   Connection for MPG input terminal signal

15.4.1   MPG terminal layout

The NC200 series controller features one manual pulse input for machine's hand wheel operation. Its internal DC 5V working power can power the hand wheel device directly. See the figure below for its wiring:
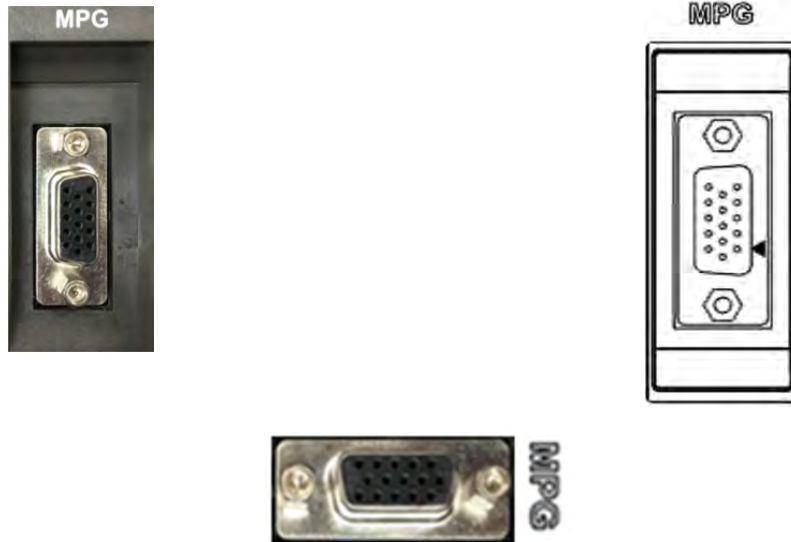
MPG	
Pin No	Function
PIN 1	External 24 V input bi-directional
PIN 2	IN1(X28) X-axis
PIN 3	IN2(X29) Y-axis
PIN 4	IN3(X30) Z-axis
PIN 5	IN4(X31) magnification x1
PIN 6	IN5(X32) magnification x10
PIN 7	IN6(X33) magnification x100
PIN 8	IN7 (X26) A-axis
PIN 9	OUT8(Y27) digital output
PIN 10	+5V_OUT
PIN 11	XA+
PIN 12	XA-
PIN 13	XB+
PIN 14	XB-
PIN 15	+5V_GND

15.4.2 Wiring of MPG

With servo connection in MPG mode, turning the hand wheel may lead to fluctuation of **VS0** value (axis selection and factor setup are required).

MPG pulse input wiring with internal power of 5VDC
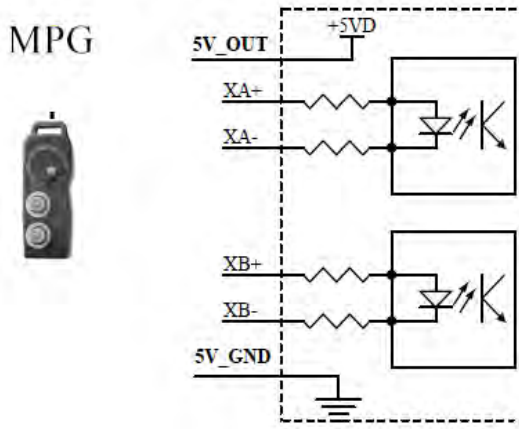
DI pin wiring
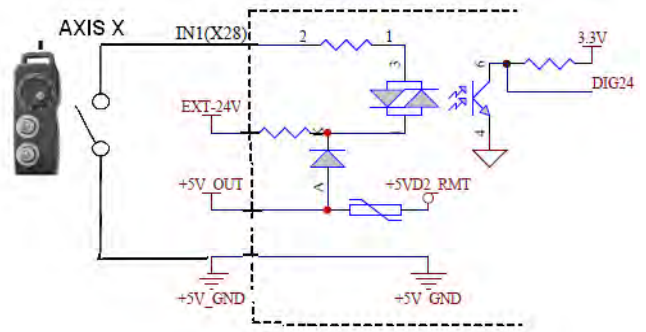
MPG Hand Wheel

For axis and triggering signal selection

Function	Special D code	Description
MPG startup operation	D1040	Set operation code for MPG hand wheel startup with value "0" for using external hand wheel and "10" for secondary control panel incremental jogging. It triggers signal M1156 and M1157.
Select the channel for MPG operation	D1041	Set to select the channel for MPG hand wheel operation. It is set to "0" now.
Status of MPG 0-axial selection knob	D1043	Set the axis to be controlled by MPG0 hand wheel: 0 for X-axis; 1 for Y-axis; 2 for Z-axis.
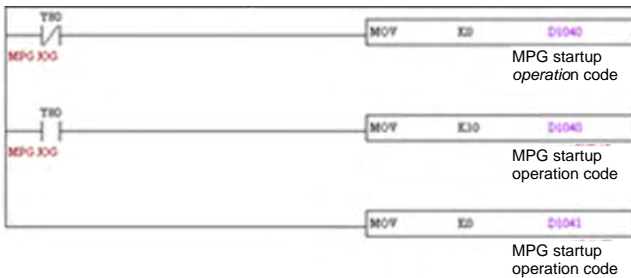
Software MPG+	M1118	Incremental jogging of secondary control panel, forward triggering signal, see D1040
Software MPG-	M1119	Incremental jogging of secondary control panel, backward triggering signal, see D1040

Magnification Adjustment (MPG)

For selecting MPG magnification in 3 stages, each stage is magnified 10 folds within a range of 0 ~ 100.

Function name	Special D code	Description
MPG0 magnification knob status	D1042	MPG0 hand wheel pulse input magnification factors of 1, 10, and 100. These factors apply to the least movement unit 0.001 mm, e.g. 1*0.001 = 0.001 mm/cnt.
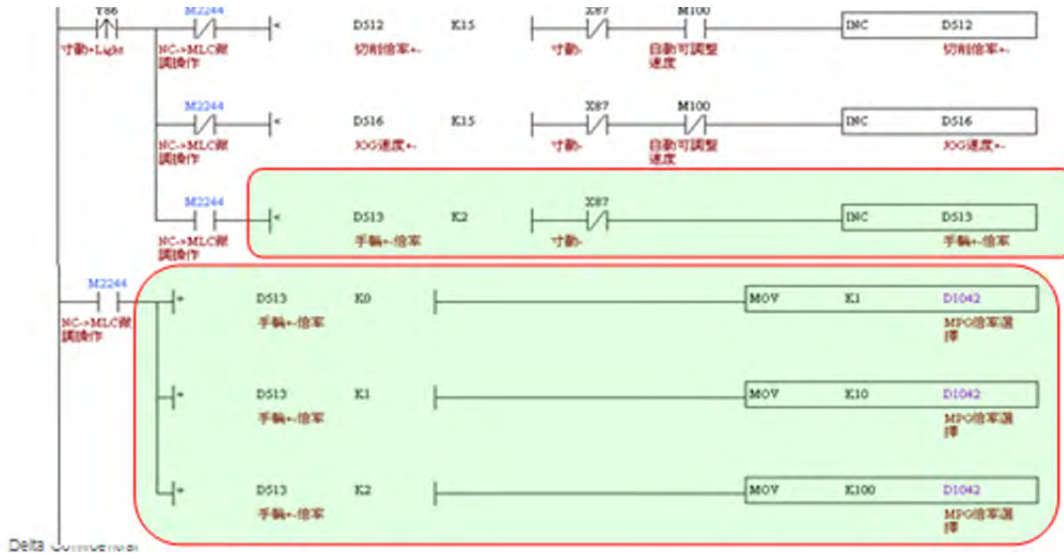
It varies with the wiring of the hand wheel (100 PPR for every cycle): Single-ended EHDW-BA6SI and differential (EHDWBE6SI). Below is the wiring of hand wheel of single-ended EHDW-BA6SI.
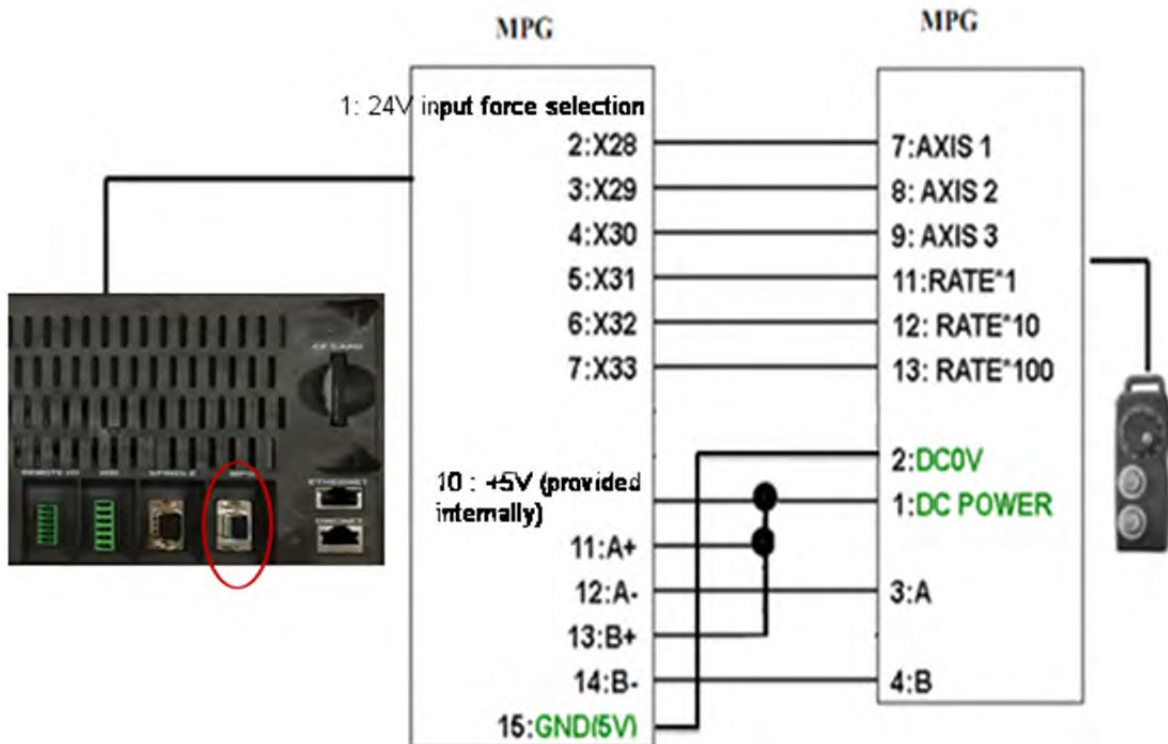
Wiring for differential EHDW-BE6SI hand wheel:
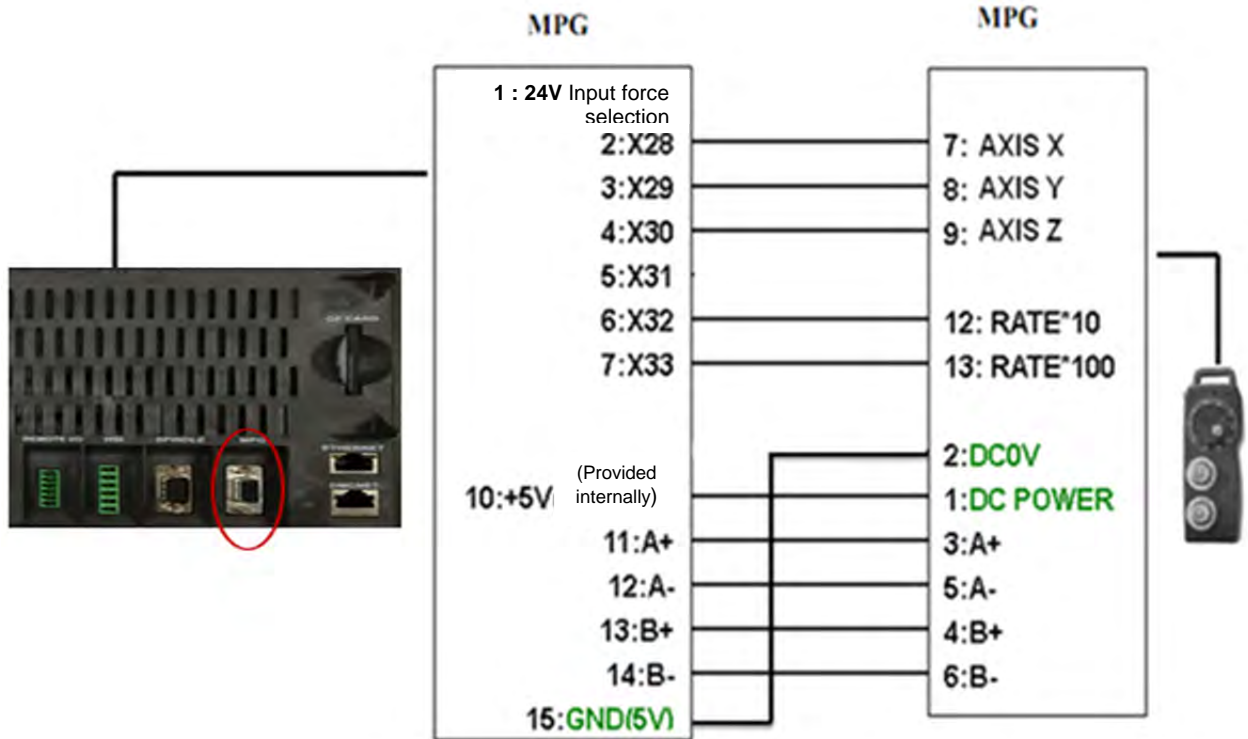
15.5 Wiring for Remote I/O Signal

15.5.1 Remote I/O terminal layout

NC200 series provide a REMOTE I/O port which can be serial connected to up to 8 expansion modules with 256 input and 256 output points. See pin definition and illustration below.
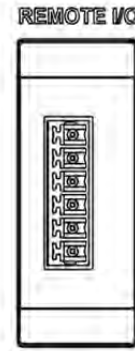
REMOTE I/O	
Pin No	Function
PIN 1	TX+
PIN 2	TX-
PIN 3	RX-
PIN 4	RX+
PIN 5	GND
PIN 6	SHIELD

15.5.2 Wiring description for Remote I/O
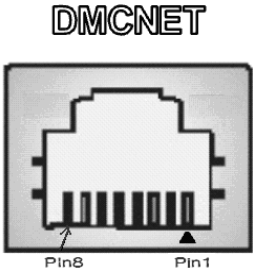
6-pin connector at NC300A wiring end	REMOTE I/O module	Connector illustration
TX+ (1) —————— (1) RX+ TX- (2) —————— 2) RX- RX- (3) —————— (3) TX- RX+ (4) —————— (4) TX+ GND (5) —————— (5) GND SHIELD (6) —————— (6) SHIELD		 Top view
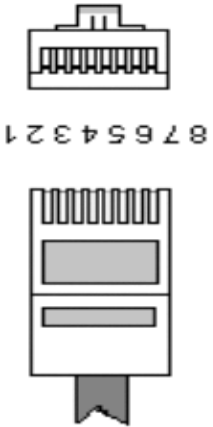
15.6 Wiring of DMCNET Connector

15.6.1 Terminal layout of DMCNET

The communication between of NC200 series controller and servo system applies DMCNET, a new, real-time network system with redundancy function provided by Delta. See pin definitions below:

Pin No	Signal	Function	Connector illustration
1	DMCNET_1A	DMCNET Channel 1 bus line (+)	
2	DMCNET_1B	DMCNET Channel 1 bus line (-)	
3	DMCNET_2A	DMCNET Channel 2 bus line (+)	
4	-	Reserved	
5	-	Reserved	
6	DMCNET_2B	DMCNET Channel 2 bus line (-)	
7	-	Reserved	
8	-	Reserved	

Wiring description:

8-pin DMCNET connector (RJ45) at the NC200 wiring end	8-pin DMCNET connector (RJ45) at the Servo wiring end	Connector illustration
Orange & white (1) ——————— (1) Orange & white		
Orange (2)——————— (2) Orange		
Green & white (3)——————— (3) Green & white		
Blue (4)——————— (4) Blue		
Blue & white (5)——————— (5) Blue & white		
Green (6)——————— (6) Green		
Brown & white (7)——————— (7) Brown & white		
Brown (8)——————— (8) Brown		Top view
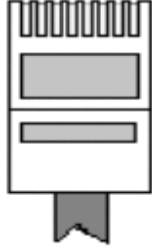
15.7 Wiring for Ethernet communication terminal

15.7.1 Ethernet connector terminal layout

The NC200 series controller reserves Ethernet channel for communication with PC or network. See the figure below for the wiring pins:
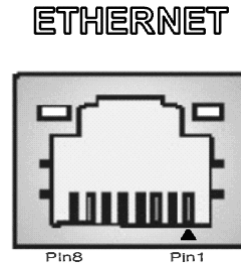
Wiring description

8-pin Ethernet connector (RJ45) at NC200 wiring end	8-pin ETHERNET connector (RJ45) at wiring end of HUB	Connector illustration
Orange & white (1) ———— (1) Orange & white		
Orange (2) ———— (2) Orange		
Green & white (3) ———— (3) Green & white		
Blue (4) ———— (4) Blue		
Blue & white (5) ———— (5) Blue & white		
Green (6) ———— (6) Green		
Brown & white (7) ———— (7) Brown & white		
Brown (8) ———— (8) Brown		Top view

15.8   Wiring of HSI

15.8.1   HSI connector terminal layout

NC200 series provide HSI terminal; see the illustration and pin definition below :

Pin definition

Pin No	Signal	Function	Connector illustration
1	HSC_COM	High speed counter input 2_GND	
2	HSC_DI2	High speed counter input 2 (9V 10mA)	
3	HSC_COM	High speed counter input 1_GND	
4	HSC_DI1	High speed counter input 1 (9V 10mA)	Top view
5	EX_GND	Emergency stop GND	
6	EX_EMG2	Emergency stop input	

The wiring of high speed counter DI supports a band up to 5 MHz input and is suitable for G31 interruption input with external power supply(voltage: 22 ~ 26 V, allowable current: 8 ~ 20 mA and surge current: below 50 mA). Setting is as follows: set high speed counter input 1 for G31 interruption with parameter 46Bit5 = 1; parameter 307Bit4 = 1; input limit parameter 25Bit0 = 1 and special M = M2142.

15.8.2   Wiring for G31

Below is the DI wiring with external power supply (Voltage: 9 V above; allowable current: 8 ~ 20 mA; surge current: under 50 mA).
For wiring of the high speed counter input (input bi-directional optical coupler), HIS_1 can be set for G31 interrupt input.
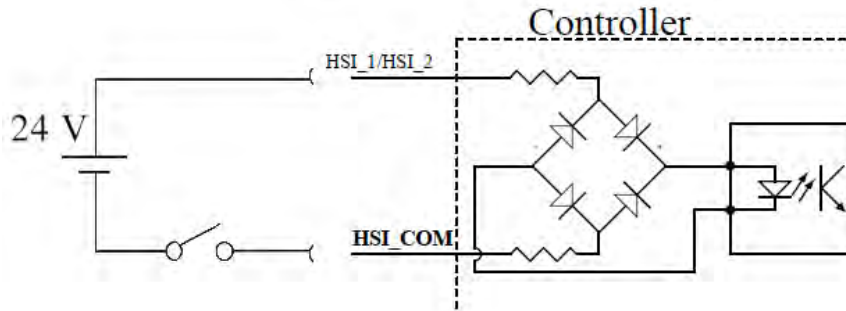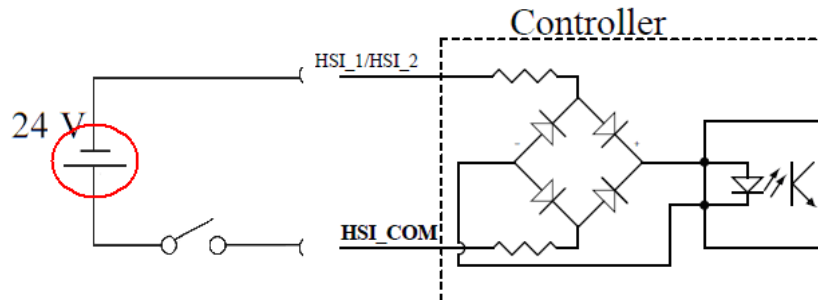
15.9 Definitions of the secondary Control Panel I/O

NC200A-MI-A (all-in-one model)
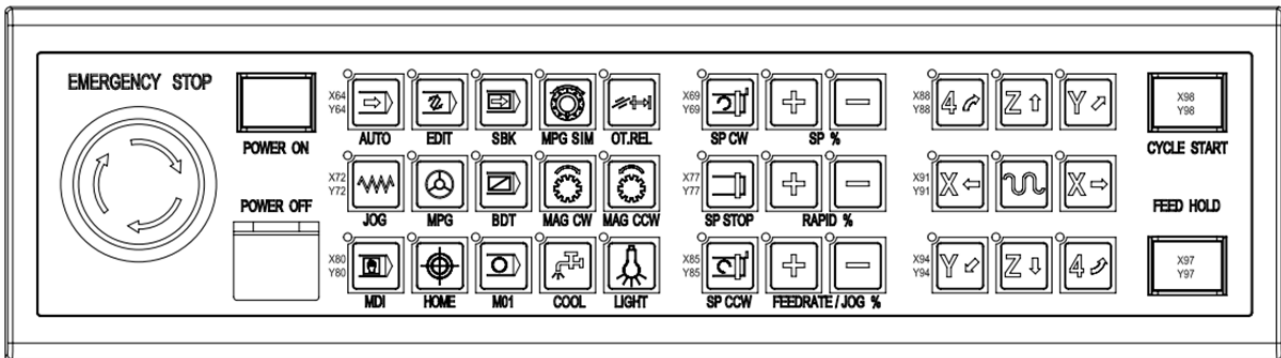
15.10 Electrical specifications

15.10.1 Electrical specifications of NC200 series

NC200A:

Model	NC200A
Working environment	10% ~ 95% RH (0 ~ +55°C)
Storage environment	10% ~ 95% RH (-20~ +60°C)
Cooling	Natural cooling
Safety Certification (Panel waterproof level)	IP65 / NEMA4 / CE, UL [Note 4]
Working voltage [Note 5]	DC +24 V (-10% ~ +15%) (embedded isolated circuit [Note 3])
Insulation endurance	Between DC24 and FG terminals: AC500V, 1 minute
Power consumption [Note 5]	24 V 0.6 A 15 W
Backup battery for memory	3 V lithium CR2032 × 1
Life span of backup battery	Varies with ambient temperature and working conditions; about three years in 25°C room temperature
Vibration/collision resistance	IEC61131-2 specifications: Intermittent vibration: 5 Hz-9 Hz 3.5 mm, 9 Hz-150 Hz 1 G Continuous vibration: 5 Hz-9 Hz 1.75 mm, 9 Hz-150 Hz 0.5 G 10 times in direction X, Y, and Z respectively
Dimensions (W) x (H) x (D) mm	400 x 309 x 123
Opening dimensions (W) x (H) mm	378 x 291
Weight	Approx. 4200 g

15.11 Appearance and dimensions

15.11.1   Appearance and dimensions of NC200 series
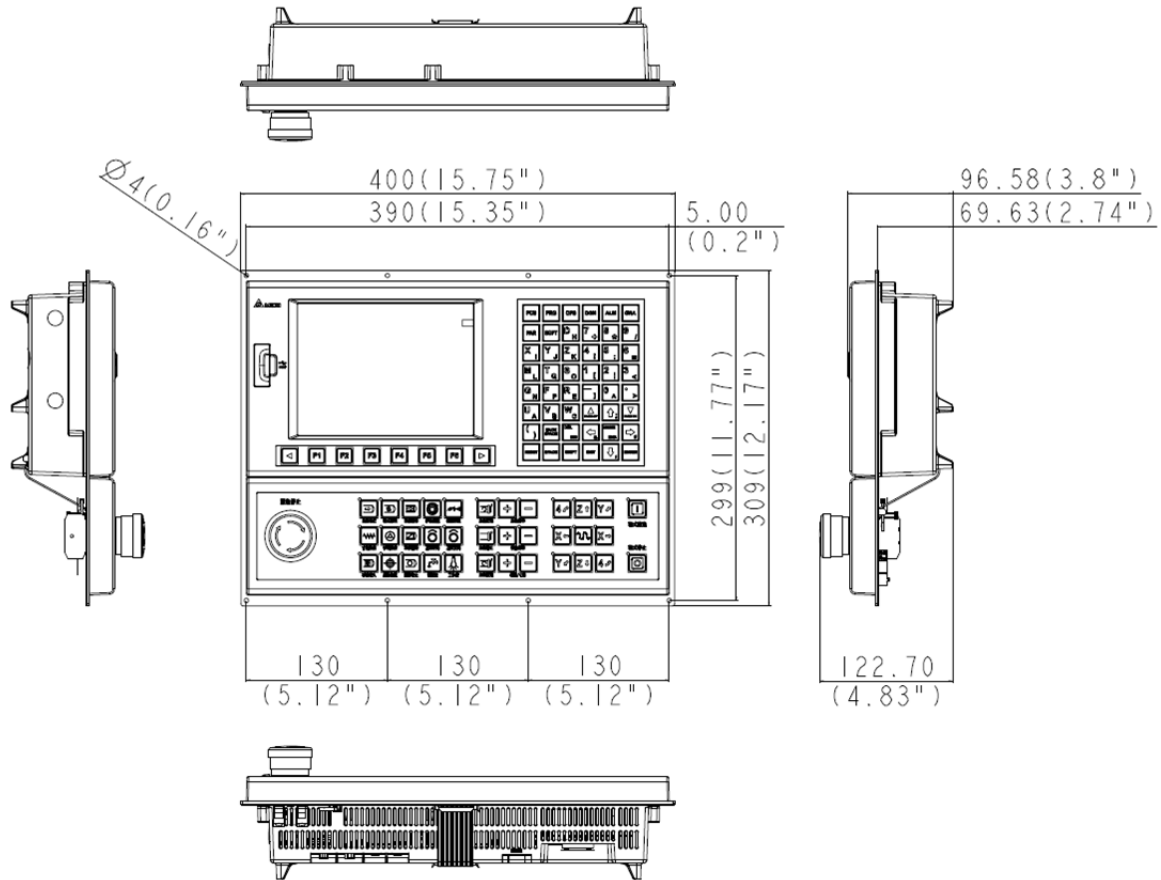
Cut-out Dimensions
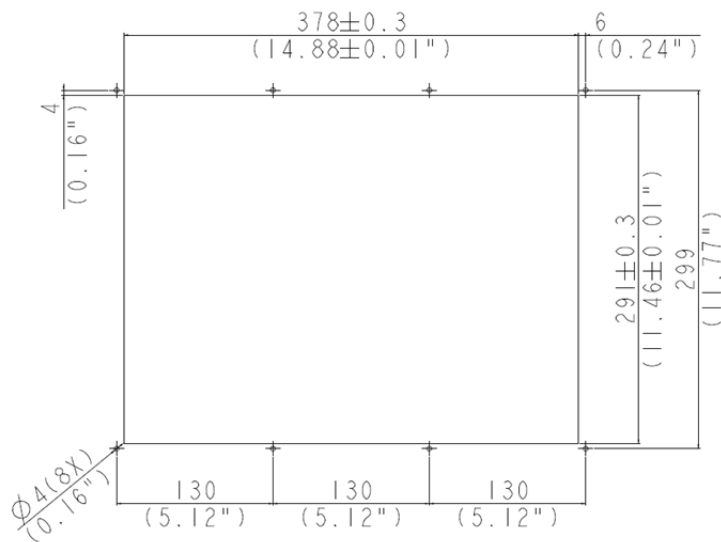
(This page is intentionally left blank.)
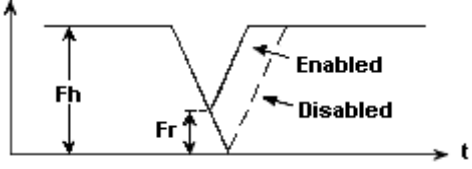
Chapter 16　Parameters

16.1 Overview

This document summarizes all information on parameters accessible to users including: parameter ID, parameters' name, description, default values, valid value range, and remarks. Please refer to this document for parameter settings.

(**P**)　Parameter values take effect only after machine is restarted
(●)　Cannot reset to default values. Changes can only be made manually
(**R**)　Parameter values take effect only by pressing the Reset key

16.2　Machining parameters

No.	Name	Description	Default	Range	Remark
309	Arc feed rate	Set up feed rate of arc with radius 2 mm.	1000	10~50000	
310	Minimum arc feed rate	Set up the minimum feed rate for executing art cutting command.	500	10~50000	P mm/min
311	Overlapped speed reduction ratio (corner speed limit)	Set the transfer speed limit between blocks; higher value means less speed change. UNIT: mm/min	100	0~50000	1
315	F0 speed	Set up speed of Rapid feed F0. UNIT: mm/min, inch/min	100	10~10000	1
316	Rapid speed (G00 feed rate)	Set up default of Rapid speed. UNIT: mm/min, inch/min	5000	1~60000	1
317	Rapid ACC/DEC time G00 acceleration/deceleration time constant	Set up acceleration time of high speed moving; its S curve time is the same as that of item 319. UNIT: 0.001 sec	200	1~2000	1
318	Maximum moving speed	Set up the maximum cutting speed. UNIT: mm/min, inch/min	5000	1~60000	1

No.	Name	Description	Default	Range	Remark
319	ACC/DEC time Cutting speed of acceleration/deceleration time constant	Set up the acceleration time for cutting speed (before ACC/DEC interpolation) UNIT: 0.001 sec (before ACC/DEC interpolation)	200	1~2000	1
320	S curve time constant (before interpolation)	Set up S curve time UNIT: 0.001 sec (before ACC/DEC interpolation)	20	1~2000	1
321	ACC/DEC time	Set up the acceleration time for Rapid speed command; The larger the value is the more significant the profiling error will be. UNIT: 0.001 sec (before ACC/DEC interpolation)	50	1~500	1
322	S curve time constant (after interpolation)	Set up the acceleration/deceleration time of S curve. UNIT: 0.001 sec (after ACC/DEC interpolation)	10	1~100	1
324	Peck retrieving amount	Set up the retrieving amount of pecking. UNIT: um	100	1~50000	
326	Cycle parameter	<table><tr><td>Bit</td><td>Description</td><td>Range</td></tr><tr><td>0</td><td>Tool direction cyclic retrieving: 0: +X 1: -X 2: +Y 3: -Y</td><td>0~3</td></tr><tr><td>2~3</td><td>Tapping mode 0: general mode 1: deep pecking (feeding amount Q, retrieve to R) 2: general pecking (feeding amount Q, retrieving amount D)</td><td>0~2</td></tr></table>	0	0~ 0xFFFF	
327	EMG stop time constant	Set up the time required for a servo motor to decelerate to stop after the emergency button is pressed.		5~500	0.001 sec
328	EMG stop delay time	Set up the delay time of flag enabled (M2115) during emergency stop under Servo Off status.	35	0~200	0.001 sec
418	Feed forward gain ratio of the master axis	Set the feed forward compensation proportion of the master axis. (not available)	0	0~200	
635	Feed forward gain ratio	Set up the feed forward compensation proportion of each axis. (not available)	0	0~200	

16.3   Operation parameters

No.	Name	Description	Default	Range	Remark
3~12	GO9010 ~ GO9019	Set G code to call macro O9010; 0: disable the function of calling macros	0	0~1000	
		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	MO9020 ~ MO9029	Set M code to call macro O9020. 0: disable the function of calling macros	0	0~1000	
		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			
13~22	MO9020 ~ MO9029	Set M code to call macro O9027	0	0~1000	
		Set M code to call macro O9028			
		Set M code to call macro O9029			
23	TO9000	Set T code to call macro O9000. 0: disable the function of calling macros 1: any T code will call macro	0	0~1	
24	RO9030	Breakpoint search 0 (Off): Continue machining from statement after the breakpoint found. 1 (On): Execute program O9030 after breakpoint is found and resume machining after the breakpoint.	0	0~1	

No.	Name	Description			Default	Range	Remark
25	System DIO signal polarity settings	Set up DI polarity of system HIS 1/2			0	0~1	
		Bit	Description	Range			
		0~1	HIS 1 (G31 interrupt) and HIS 2 settings Bit0: HIS1 input Bit1: HIS2 input 0: b contact open->H, close->L 1: a contact open->L, close->H	0~3			
46	System utility settings	Set up system utility procedures			1100	0~0xFFFF	
		Bit	Description	Range			
		5	High speed input point (G31) 1: enable input point of G31 0: Disable	0~1			
		10	Ignore movement command floating point 0: Do not ignore, i.e. 1 = 1 µm 1: Ignore, 1 = 1 mm	0~1			
		12	Macro preview function 0: Disable 1: Enable	0~1			
47	MPG hand wheel gain	MPG filter gains for hand wheel effect adjustment. The larger the value is, the faster the system reacts and the more it vibrates. UNIT: 0.0001			100	1~60000	1
48	MPG hand wheel filter	MPG filter settings: 0: None Level: 1, 2, 3, 4, 5, 6 Khz: 312, 10, 5, 2.5, 1.6, 1.2			0	0 ~ 6	
49	Servo axis output settings	Port configuration for the 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.			0	0 ~ 1	
50	Show macro file	Bit | Description | Range 0 | Show macro file O | 0 ~ 1 1 | Show G/M macro file | 0 ~ 1			0	0 ~ 3	
301	Unit paramter	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.			3	0~4	
306	G code function setup	Unit: 0: metric; 1:inch G21 for inputting metric or G20 for inch			0	0~1	

		Bit	Description	Range			
307	Channel utility setup	4~5	G31 input selection 0: input via PLC 1: HIS 0 (latch input 1) 2: HIS 1 (latch input 2)	0~2			
		6	Continue the execution 0: During the execution of one single block: offset, move remaining coordinates, and recover positions in next block when running the program again; see illustration 1: During the execution of one single block: offset, when running the program again, recover the offset and move remaining coordinates; see illustration	0~1			
		8~9	Emergency stop mode 0: emergency stop then Servo Off 1: emergency stop then Servo Off after some delay 2: emergency stop without Servo Off	0~2			
350 ~ 359	Halt M code 1 ~10	Halt M code 1 (0: no setting) Halt M code 2 Halt M code 3 Halt M code 4 Halt M code 5 Halt M code 6 Halt M code 7 Halt M code 8 Halt M code 9 Halt M code 10		0	0~1000	**P**
360	Synchronous control direction	Synchronous control direction: Bit0 ~ 5: Synchronous control X ~ C 0: same direction 1: different direction		0	0 ~ 5	
361 ~ 366	Synchronous control X ~ C	Slave axis X follows the master axis 0: close 1 ~ 6: X ~ C Slave axis Y follows the master axis Slave axis Z follows the master axis		0	0 ~ 6	**P**

		Slave axis A follows the master axis			
		Slave axis B follows the master axis			
		Slave axis C follows the master axis			
370	Transfer control direction	Transfer control direction Bit0~5: Synchronous control X ~ C 0: same direction 1: different direction	0		**P**
371 **~** **376**	Transfer control X~ C	Axis X transfers the command of master axis. The master axis does not move at the moment. 0: close 1~6: X ~ C	0	1 ~ 6	**P**
		Axis Y transfers the command of master axis.			
		Axis Z transfers the command of master axis.			
		Axis A transfers the command of master axis.			
		Axis Btransfers the command of master axis.			
		Axis C transfers the command of master axis.			
2010	High speed input trigger settings	<table><tr><th>Bit</th><th>Description</th><th>Range</th></tr><tr><td>0</td><td>HSI 0 trigger settings</td><td>0~1</td></tr><tr><td>1</td><td>HSI 1 trigger settings</td><td>0~1</td></tr></table>Set up rising edge (set to 0) counting for high speed input. Set up falling edge (set to 1) counting	0	0 ~ 65535	P
621	Maximum allowable speed	Set up maximum axial moving speed in unit of mm/min or inch/min	5000	0 ~ 60000	mm/min inch/min rpm
622	ACC / DEC time constant	Set up acceleration time for manual and homing operation	50	0~10000	0.001 sec
623	S curve time constant	Set up S-curve time constant for manual and homing operation	50	0~10000	0.001 sec
624	EMG axial DEC time	Time constant for emergency stop of the axis in Jog operantion	100	1 ~ 2000	0.001 sec
643	Allowable following error	If the following error exceeds the setting value when moving, the alarm occurs. (CU: command unit)	30000	1~32767	CU

16.4   Tool magazine parameters

No.	Name	Description			Default	Range	Remark
304	Tool magazine parameters setup	Tool magazine selection			0	0	**P**
		Bit	Description	Range			
		14	Enable ATC (Automatic Tool Change) function 0: Off 1: On	0~1			
		9	Set up ATC number 0: tool magazine 0 1: tool magazine 1	0~1			
		11	ATC type 0: switching (not available) 1: non switching	0~1			
		12~13	Set up ATC searching mode 0: shortest length path 1: CW 2: CCW	0~2			
		15	Control type (not available) 0: MLC control 1: NC control	0~1			
336	Tool magazine database control	Bit	Description	Range	0x0800	0~65535	**P**
		11	ATC type (not available) 0: Exchanger type (switching arm) 1: Non-exchanger type (tool tray)	0~1			
337	Database control for tool magazine	Bit	Description	Range	1	0~3	
		0	Enable ATC 1 0: Disable 1: Enable	0~1			
		1	Enable ATC 2 0: Disable 1: Enable	0~1			
338	ATC 1 station	Set up number of stations of tool magazine 1.			10	2~255	
339	ATC 1 init number	Set up tool number after tool magazine 1 is reset			1	1~100	
340	ATC 1 start number	Set up starting number of tool magazine 1			1	1~100	

341	ATC 2 station	Set up number of stations of tool magazine 2.	10	2~255	
342	ATC 2 init number	Set up tool number after tool magazine 2 is reset	1	1~100	
343	ATC 2 start number	Set up starting number of tool magazine 2	1	1~100	

16.5   Spindle parameters

No.	Name	Description			Default	Range	Remark
		Bit	Description	Range			
399	Spindle mode	0	Spindle function on/off 0: Spindle Off 1: Spindle On	0~1			
		1	Close loop control flag 0: Close loop control Off 1: Close loop control On (requires feedback encoder)	0~1			
		2~3	Spindle output mode 0: DMCNET (servo spindle) 2: EDAC (analog output)	0~2			
		4	Speed mode 0: rmp / 1: PPM	0~1			
		5	Selection for spindle encoder type 0: high resolution (x1000) 1: normal resolution (x4)	0~1			
401	Spindle input port number	Set up feedback channel of spindle port for the encoder. (switching function not available yet)			8	0~8	
402	Pulse number of spindle encoder	Set up the pulse number of encoder.			1280	2~10000	pulse/ rev
403	Spindle gains	Set up speed regulator gains (the smaller the value is the faster the response will be).			50	1~5000	
404	Spindle positioning speed	Set up positioning speed of the spindle.			100	1~20000	rpm
405	Spindle positioning offset	Set up positioning offset of the servo spindle.			0	0~36000	0.01 degree
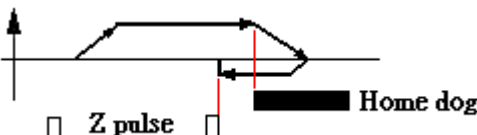
406	Spindle target speed error	Set up tolerance between spindle's nominal and actual speeds.	10	0~100	
407	Spindle positioning error	Set up spindle positioning error.	100	0~36000	0.01 degree
408	Range of spindle zero speed	The zero speed signal of spindle will be on when its speed is within the setting range **(NC-> MLC M2257).**	5	0~1000	rpm
409	Spindle maximum speed	Set up maximum speed of the spindle. UNIT: rpm	20000	0~50000	1
410	Spindle minimum speed	Execute rigit tapping when spindle operates at minimum speed. UNIT: rpm	10	0~10000	1
411	Spindle acceleration time	Set up the acceleration time of the spindle of fast speed. UNIT: 0.001 sec	20	1~2000	1
412	Spindle S curve time constant	Set up S curve time of the spindle. UNIT: 0.001 sec	10	1~2000	1
416	Tapping acceleration/deceleration time constant	Set up spindle acceleration / deceleration time for tapping	2000	1~20000	0.001 sec
417	Tapping S curve time constant	Set up spindle S curve time for tapping	100	1~2000	0.001 sec
420	Spindle positioning low speed	Low speed positioning of the spindle [Unit: rpm]	100	1~20000	rpm
421	Spindle retrieve ratio	The spindle retrieve ratio. The setting speed is the maximum, e.g. tapping speed is S1000; if the parameter is set to 20, retrieve speed will be S2000. (Uint: 0.1)	10	10 ~ 50000	
422	Gear ratio numerator 1	Set the numerator of the spindle gear ratio (speed at first gear)	1	0~60000	1
423	Gear ratio denominator 1	Set the denominator of the spindle gear ratio (speed at first gear)	1	0~60000	1
424	Gear ratio numerator 2	Set the numerator of the spindle gear ratio (speed at second gear)	1	0~60000	1
425	Gear ratio denominator 2	Set the denominator of the spindle gear ratio (speed at second gear)	1	0~60000	1
426	Gear ratio numerator 3	Set the numerator of the spindle gear ratio (speed at third gear)	1	0~60000	1
427	Gear ratio denominator 3	Set the denominator of the spindle gear ratio (speed at third gear)	1	0~60000	1
428	Gear ratio numerator 4	Set the numerator of the spindle gear ratio (speed at fourth gear)	1	0~60000	1
429	Gear ratio denominator 4	Set the denominator of the spindle gear ratio (speed at fourth gear)	1	0~60000	1

16.6   Mechanical parameters
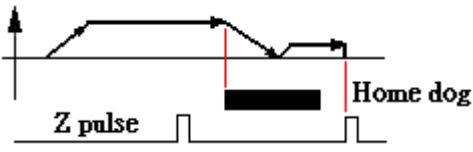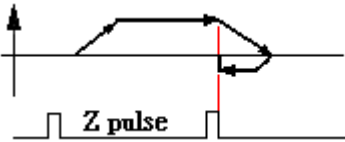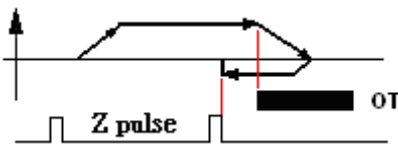
No.	Item	Description	Default	Range	Length (word)
602	First positive soft limit	Set up positive software limit. 0 = OFF Positive software limit alarm occurs when exceeding the limit range. Can be controlled by special M (set M1250 to NO to cancel first software limit) UNIT: um	10^8	-10^8 ~ $+10^8$	2
603	First negative soft limit	Set up negative software limit. 0 = Off Negative software limit alarm occurs when exceeding the limit range Can be controlled by special M UNIT: um	-10^8	-10^8 ~ $+10^8$	2
604	Second positive soft limit	Set up positive software limit. 0 = Off Positive software limit alarm occurs when exceeding the limit range	10^8	-10^8 ~ $+10^8$	2
605	Second negative soft limit	Set up negative software limit. 0 = Off Negative software limit alarm occurs when exceeding the limit range UNIT: um	-10^8	-10^8 ~ $+10^8$	2
628	Polarity of hard limit setting	Set up forward/backward hardware limit and home input polarity. Value 0 = input by Hi activity and on/off at A connector 0 = input by Lo activity and on/off at B connector 	Bit	Description	Range
0	positive limit	0~1
1	negative limit	0~1
2	home	0~1		0		
630	Encoder pulse number	The pulse number in each revolution of the motor when ASDA-A2 is employed (default)	1280	10 ~ 50000	
631	Number of teeth of spindle	Set up teeth number at the transmission shaft end	1	1~65535	
632	Number of teeth of motor	Set up teeth number at the motor end	1	1~65535	
633	Lead screw pitch	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 linear axis.	10	2~100	mm

634	Axis control variables	Bit	Description	Range	5	0~65535	2
		1~ 2	Feed mode of the rotation axis is only suitable for axis A, B and C. Axis X, Y and Z is not applicable. 0: the rotation axis is not the shortest path 1: the rotation axis is the shortest path 2: path of the rotation axis is displayed in linear format 3 ~ 4: reserved 5: linear axis	0~5			

16.7   Homing origin parameters

No.	Item	Description	Default	Range	Length (word)
606	Mechanical origin coordinates (offset)	Set up of absolute origin of mechanical coordinates. If this value is modified, this modified value will be referred when homing. UNIT: CU (float value)	0	-10^8 ~ 10^8	
607	Second reference coordinates	Ccoordinates of the second reference point (G30 position setup) UNIT: CU (float value)	0	-10^8 ~ 10^8	
608	Third reference coordinates	Coordinates of the third reference point UNIT: CU (float value)	0	-10^8 ~ 10^8	
609	Fourth reference coordinates	Coordinates of the fourth reference point UNIT: CU (float value)	0	-10^8 ~ 10^8	
610	Second reference position error settings	Error settings of the second reference position, e.g. value 0.2 indicates that when +-0.2 mm section's second reference origin arrived, M2288 will be On.	0	-10^8 ~ 10^8	
616	Homing mode	0: Homing disabled 1: Mode 1 When touching the block, move in reverse direction to look for Z pulse and regard it as the origin. 	1	0~5	1
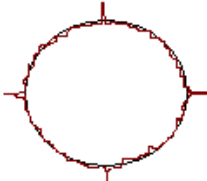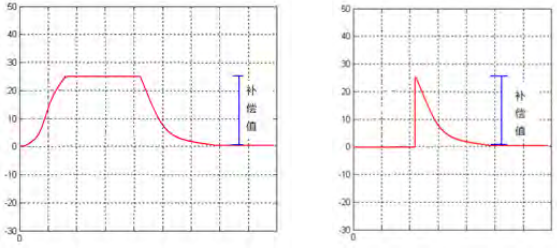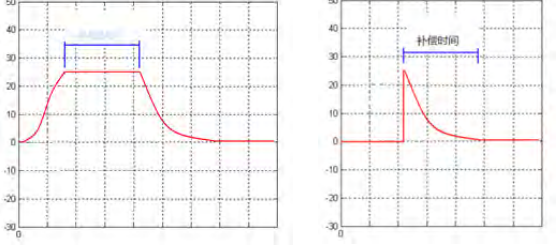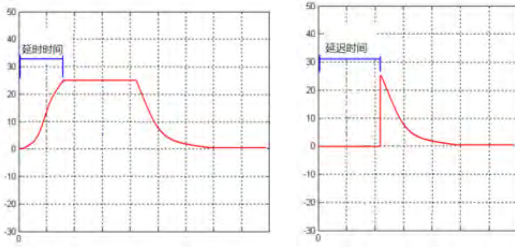
		2：Mode 2 When touching the block, move in forward direction to look for Z pulse and regard it as the origin. 3：Mode 3 Move to Z pulse at low speed. 4：OT mode Regard the software limit as home sensor in homing mode; regard the software limit as limit block in other modes. 5：Mode for absolute motor			
617	Searching the origin	Bit 0 : Direction searching mode for returning to the origin 0: clockwise (forward) 1: counterclockwise (backward)	1	0~7h	1
618	First stage speed of homing	Set up the starting speed for searching the home sensor. UNIT: mm/min	2000	0~10000	1
619	Second stage speed of homing	Set up the speed for searching Z pulse. UNIT: mm/min	200	0~2000	1
620	Reference point movement speed	Set up feed rate for moving from the first reference point to the origin UNIT: rpm	10	0~20000	1

16.8   Network parameters

No.	Item	Description	Default	Range
10030	Host name	Host name	CNC000	1~8
10031	IP Address	System IP address	0.0.0.0	0~255
10032	Subnet mask	System subnet mask	0.0.0.0	0~255
10033	Default gateway	System default gateway	0.0.0.0	0~255
10034	Enable Ethernet function	Enable system network function 0: Off 1: On	0	0~1
10035	Enable DHCP function	Enable DHCP function 0: Off 1: On	0	0~1
10036 ~ 10040	Remote PC IP Address 1 ~ 5	IP address 1 IP address 2 IP address 3 IP address 4 IP address 5	0	0~255
10041	IP address 1 for remote directory sharing	Edit IP address of computer defined by Network in directory. 0: Off	0	0~5

16.9 Compensation parameters

No.	Item	Description	Default	Range
1000	Rear gap compensation value	Set up the backlash amount that is fixed in most screws of the mechanical system. There is no difference in G00 and G01 modes. 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	0~10.0
1001	Backlash compensation time	Set the time constant for compensation ratio for the movement direction of backlash compensation. It is effective only when rear backlash compensation value is given. UNIT: 0.0001 sec	0	0~10000
1002	Backlash compensation delay time	Set up time delay for startup compensation. UNIT: 0.0001 sec	0	0~10000
1003	Friction compensation amount	Friction compensation amount (mm) Friction compensation mode = 0 Friction compensation mode = 1	0	0~10.0
1004	Friction compensation time	Friction compensation time (0.0001 sec) Friction compensation mode = 0 Friction compensation mode = 1	0	0~10000
1005	Friction compensation delay time	Friction compensation delay time (0.0001 sec)	0	0~10000

		Friction compensation mode = 0 Friction compensation mode = 1			

		Bit	Description	Range		
1006	Pitch compensation application setup	0	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)	0~1		
		2	Make friction compensation when machine tool moves in forward direction.	0~1		
		3	Make friction compensation when machine tool moves in reverse direction.	0~1		
		4	Mode selection for friction compensation	0~1		
		15	LSC direction; Direction of the mechanical coordinates (Measuring direction from starting point) 0: measuring toward forward direction 1: measuring toward backward direction	0~1		
1007	Measuring points	Set up the correction value in screw compensation table. If the value is set to zero, compensation is turned off.			0	0~128
1008	Measuring intervals	Set up interval for screw compensation. UNIT: mm			Float value	1~300
1009	Measuring offsets	Set up the offset amount from homing origin to the measuring point. Comply with direction of Bit 15 of parameter 1006.			Float value	-1000 ~ 1000
1010 ~ 1137	Data 1 ~ data 128	1st ~128th screw compensation value; the first point is set to the origin UNIT: linear axis (mm); roation axis (deg)			Float value	-20~20

16.10 System parameters

No.	Name	Description	Default	Range
10000	System date	Set up system date Format: yyyy/mm/dd	0	
10001	System time	Set up system time Format: hh:mm:ss	0	
10002	System language	System multi-language 0: English 1: Chinese	0	0~1
10003	Screen brightness	Set up screen brightness	0	0~60
10004	User defined language	User defined system language	0	0~10
10009	Synchronous coordinate display	Method of showing the synchronous coordinate	0	0~2
10010	Screen saver ON	Activate screen save function 0: On 1: Off	0	0~1
10011	Time of first stage screen saver	When screen saver is On, set up the time of first stage screen saver.	0	0~60
10012	Brightness of first stage screen saver	When screen saver is On, set up the brightness of first stage screen saver.	30	0~60
10013	Time of second stage screen saver	When screen saver is On, set up the time of second stage screen saver.	30	1~60
10014	Brightness of second stage screen saver	When screen saver is On, set up the brightness of second stage screen saver.	10	0~60

No.	Name	Description			Default	Range
10016	System related setting	System related setting			0	0~10.0
		Bit	Description	Ragne		
		0	Reset system after EMG released Automatically generate a Reset signal after releasing EMG 0: Off / 1: On	0~1		
		1	Displaying software panel when startup as default 0: Off 1: Display the software panel at startup	0~1		
		2	Pop up the alarm screen 0: Not pop up 1: Pop up the alarm screen when an alarm occurs	0~1		
		3	Lock the group key at software panel	0~1		

		Bit	Description	Ragne		
			0: Unlock 1: Lock (When the function of Displaying software panel at startup is set to 1, this function will be locked.)			
10017	Open G code editor	0	Enable G code editing Enable G code editing 0: Off 1: On	0~1	0	0~10000
		1	Call MACRO source file 0: CF card 1: INTER memory	0~1		
		3	Enable feed speed setup 0: Disable 1: Enable	0~1		
		4	Program reset after editing 0: Disable 1: Enable	0~1		
10018	Background color	Background color			LIGHT GRAY	0~65535
10019	Title color	Text color in the title bar			BLACK	0~65535
10020	Mode color	Text color in the mode bar			DARK BLUE	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	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	Highlited text color of software panel	Highlited text color of software panel			COLOR_S07	0~65535

16.11 MLC parameters

No.	Item	Description	Default	Range
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	0~60
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	LIGHTBLUE	0~65535
12009	Monitoring color of ladder diagram	Monitoring color in the ladder diagram	LIGHTGREEN	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	MLC editing protection 0: protection off 1: protection on	1	0~1

16.12   Graphic parameters

Item	Name	Description	Default	Range
14000	Line color	Line color of the graph	BLACK	0 ~ 65535
14001	Background color	Background color of the graph	SEA	0 ~ 65535
14002	Line width	Line width of the graph	1	0 ~ 4
14003	Workpiece width	Workpiece width in the graph (mm)	200	0 ~ 10000
14004	Workpiece height	Workpiece height of in the graph (mm)	200	0 ~ 10000
14005	Workpiece length	Workpiece length in the graph (mm)	200	0 ~ 10000
14006	Workpiece X offset	Moving distance on the X-axis of the workpiece in the graph	100	-10000 ~ 10000
14007	Workpiece Y offset	Moving distance on the Y-axis of the workpiece in the graph	100	-10000 ~ 10000
14008	Workpiece Z offset	Moving distance on the Z-axis of the workpiece in the graph	100	-10000 ~ 10000

16.13 Servo parameters

Group	No.	Name	Description	Unit	Default	Range
P1	1	Input Setting of Control Mode and Control Command	Input setting of control modes and control commands		b	0x00 ~ 0x110F (HEX)
P1	8	Smooth Constant of Position Command (Low-pass Filter)	Smooth constant of postion command	10 msec	0	0~1000
P1	36	Acceleration / Deceleration Constant of S-Curve	Acceleration and deceleration constant of S-Curve	msec	0	0~65500
P1	44	Gear Ratio (Numerator) (N1)	Set up multiple sets of gear ratio numerator	pulse	1	$1 \sim (2^{29}-1)$
P1	45	Gear Ratio (Denominator) (M)	Denominator of gear ratio (M)	pulse	1	$1 \sim (2^{31}-1)$
P1	55	Maximum Speed Limit	The maximum motor speed. Its default is set to rated speed.	r/min	0	0~65535
P1	62	Friction Compensation (%)	Level of friction compensation	%	0	0~100
P1	63	Friction Compensation (ms)	Set up the constant of friction compensation	ms	0	0~1000
P1	68	Position Command Moving Filter	Position command moving Filter	ms	4	0~100
P2	0	Position Loop Gain	Position loop gain	rad/s	35	0~2047
P2	1	Switching Rate of Position Loop Gain	Switching rate of position loop gain	%	100	10~500
P2	2	Position Feed Forward Gain	Position feed forward gain	%	50	0~100
P2	3	Smooth Constant of Position Feed Forward Gain	Smooth constant of position feed forward gain	msec	5	2~100
P2	4	Speed Loop Gain	Speed loop gain	rad/s	500	0~8191
P2	5	Switching Rate of Speed Loop Gain	Switching rate of speed loop gain	%	100	10~500
P2	6	Speed Integral Compensation	Speed integral compensation	rad/s	100	0~1023
P2	7	Speed Feed Forward Gain	Speed feed forward gain	%	0	0~100
P2	9	DI Debouncing Time	Set the time for DI debouncing time	2 msec	2	0~20
P2	23	Resonance Suppression (1)	The first setting value of resonance frequency	Hz	1000	50~1000

Group	No.	Name	Description	Unit	Default	Range
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.	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.	0.1 msec	2	0~1000
P2	26	Anti-interference Gain	The gain against external interference	0.001	0	0 ~ 1023
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	10 msec	10	0 ~ 1000
P2	43	Resonance Suppression (2)	The second setting value of resonance frequency.	Hz	1000	50~2000
P2	44	Resonance Suppression (2)	The second resonance suppression (notch filter) attenuation rate. When this parameter is set to 0, the function of Notch filter is disabled.	dB	0	0 ~ 32
P2	45	Resonance Suppression (3)	The third group of mechanism resonance frequency setting value. When this parameter is set to 0, this function will be disabled.	Hz	1000	50~2000
P2	46	Resonance Suppression Attenuation Rate (3)	The third group of resonance suppression (Notch filter) attenuation rate. Set the value to 0 to disable the function of Notch filter.	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	sec	0	0~1F
P4	0	Fault Record (N)	The lastest fault record		0	
P4	1	Fault Record (N-1)	The last second fault record		0	
P4	2	Fault Record (N-2)	The last third fault record		0	
P4	3	Fault Record (N-3)	The last fourth fault record		0	
P4	4	Fault Record (N-4)	The last fifth fault record		0	
P5	00	Fimware sub-verision	Firmware sub-version of the servo drive			

16.14   RIO Setting

RIO Setting: Press **OK** after the setting is completed.

RIO Status	Enable	RIO type	Port polarity	Disc.
1 ON	V	3	00000000	
Use the RIO station number to show the connection status, M2832 is station No.0.	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)		The setting of DI port polarity, which only can be set to station No.0 and can set DI0~DI31 (32 points in total).	Check this item means DO is on when disconnection.

Origin limit setting: Press OK after the setting is completed.

Home Limit	Set the positive / negative/ origin DI via the DI setting of RIO. This is applicable for RIO station number 0 only.		
CH0 X V Y V Z ☐ A V B V C ☐ U ☐ V ☐ W ☐	Axis selection. If none of them is selected, it means the positive, negative and origin DI is 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
Filter level 2	Y positive limit M2148	Y negative limit M2149	Y origin signal M2150
	Z positive limit M2152	Z negative limit M2153	Z origin signal M2154
	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.		

16.15   Setting for absolute motor

1. To use the CNC controller with absolute encoder, the parameter 616 (homing mode) of the corresponding connection axis must be set to 5. (Incremental and absolute encoders can be used together. If it is the first time you install absolute motor, please re-start the servo and controller after installation.)

2. After parameter setup, execute homing procedure for the absolute encoder at the pages of *DGN -> System Monitoring -> Servo Monitoring*.

3. Homing procedure has to be done only in **JOG** or **MPG** mode. Move the mechanical position of the axis with JOG or MPG operation to the proper location, press key [**1**] and then press the **Enter** key to finish homing. Them, the status indicator of origin will light up; the mechanical coordinates reset to 0, and the axis returns to the origin.

Note: When setting in MPG mode, it is effective only when axis MPG is selected. If select axis X of MPG, enter [1] and press [Enter], the homing for absolute encoder is completed.

If the alarm occurs after the homing, reset the absolute reset flag.
See below for reset alarms:

AL 060	The absolute position is lost, homing procedure is needed.
AL 061	Battery undervoltage, please change the battery.
AL 069	Wrong motor type, please make sure the absolute type encoder is connected.

(This page is intentionally left blank.)

Chapter 17 Troubleshooting

17.1 PLC system alarms (Code: 0x1200 ~ 0x1300)

Alarm code	Name	Causes and troubleshooting
0x1200	Accessing NC memory in error	An error has occurred when MLC accessing NC memory. Restart the controller or send it back for servicing.
0x1201	System is not ready	The startup procedure of NC system is not complete. Restart the controller or send it back for servicing.
0x1202	Buffer memory error	NC Buffer memory error or data is not ready. Restart the controller or send it back for servicing.
0x1203	Output port not exist	NC output port does not exist. Make sure the setting for axial parameter is correct.
0x1204	MLC code clearance error	Fail to clear the code of MLC program Please send the controller back for servicing.
0x1205	MLC flash memory error	An error has occurred when writing in MLC code to the flash memory. Restart he controller and send it back for servicing.
0x1206	SRAM error	An error has occurred when writing in SRAM. Please send the controller back for servicing.
0x1207	Host I/O channel error	An error has occurred when accessing host I/O. Please restart the controller or send it back for servicing.
0x1208	Remote I/O channel error	An error has occurred when accessing remote I/O. Restart the controller or send it back to the distributor.
0x120A	NC PAR error	NC parameter is not set up or initialized. Please re-initialize the parameter.
0x120B	Compen. PAR error	Compensation parameter memory write-in error. Please reload the compensation parameters.
0x120C	Compen. PAR clearance error	An error has occurred when clearing the compensation parameter memory. Please reload the compensation parameters.
0x120D	Compen. PAR write-in error	An error has occurred when writing in compensation parameter memory. Please reload the compensation parameters.

Alarm code	Name	Causes and troubleshooting
0x120E	PAR initialization error	An error has occurred when initializing parameters. Please re-initialize parameters.
0x120F	Memory clearance error	Unable to clear the memory normally. Please restart the controller or send it back for servicing.
0x1210	Memory write-in error	An error has occurred when writing in or initializing the memory. Please restart the controller or send it back for servicing.
0x1211	Servo is not found	Wrong parameter setting. Please verify the parameter settings
0x1212	Servo PAR type error	Wrong parameter setting; please verify the parameter setting and update the firmware.
0x1213	DMCNET initialization error	DMCNET initialization error. Please ensure the connection of DMCNET is securely connected.
0x1214	Non-volatile memory error	An error has occurred when writing in and reading the non-volatile memory. Please restart the controller or send it back for servicing.
0x1216	MLC PRG error	Check the MLC program; please reload the MLC program.
0x1217	MLC PAR setting error	Modify MLC parameter settings.
0x1300	Network com. error	Check the network connection; restart the controller or sent it back for servicing.
0x1E00	Servo error	A servo alarm has occurred. Please check the servo status or replace a new servo drive.
0x1F00	Remote I/O error	Make sure the connection for remote I/O is well or replace a new remote I/O board.

17.2   NC alarms (Code: 0x4200 ~ 0x4300)

Alarm code	Name	Causes and troubleshooting
0x4200	Homing is required	Homing is not executed yet, please execute homing procedure; check the wiring and parameter settings.
0x4201	Absolute origin unset/ loss	Set the origin position; check the battery power
0x4300	MLC accessing NC data in error	MLC is not ready or an error occurs when accessing the memory. Please restart the controller or send it back for servicing.
0x4301	MLC is not ready	MLC is not ready. Please restart the controller or send it back for servicing.
0x4302	I/O module PRG clearance error	I/O module program clearance error. Please restart the controller or send it back for servicing.
0x4303	I/O module PRG write-in error	I/O module program write-in error Please restart the controller or send it back for servicing.
0x4304	NC system PRG clearance failed	Fail to clear the NC system program. Please re-install the program.
0x4305	NC system PRG installation failed	Fail to install the NC system program. Please re-install the program.
0x4308	G code loading error	An error has occurred when loading G code. Please verify the machining program.
0x4310	I/O module PRG uninitialized	I/O module program is not initialized. Re-install I/O module program.
0x4311	I/O module initialized data length error	I/O module memory error. Please restart the controller or send it back for servicing.
0x4312	I/O module data content error	I/O module memory error. Please restart the controller or send it back for servicing.
0x4313	I/O module status error	I/O module status write-in error. Please restart the controller or send it back for servicing.
0x4314	I/O module PRG config. error	Program configuration of I/O module is in error. Please restart the controller or send it back for servicing.
0x4315	I/O module serial interface error	Reading and write-in of I/O module interface is in error. Please restart the controller or send it back for servicing.
0x4316	I/O module interface access error	An error has occurred when reading the I/O board interface. Please send the controller back for

Alarm code	Name	Causes and troubleshooting
		servicing.
0x4317	NC system cmd error	An error has occurred when reading and writing NC system command. Please send the controller back for servicing.
0x4318	NC PAR loading failure	NC parameter error or MLC is not ready. Please restart the controller or send it back for servicing.
0x4319	NC PAR does not exist	NC parameter does not exist or MLC is not ready. Please restart the controller or send it back for servicing.
0x431A	Tool magazine axis error	Tool magazine axis is not defined or defined repeatedly. Please verify the parameter setting.
0x431B	NC PAR error	NC parameter error or MLC is not ready. Please restart the controller or send it back for servicing.

17.3   Channel alarms (Code: 0xA000 ~ 0xD000)

Description of illegal G code line number and error messages

Alarm code	Name	Causes and troubleshooting
0xB003	Invalid name of the loaded file	Rename the file and press the reset key to reload the file.
0xB005	Workpiece coord. clearance error	Computing error of workpeice coordinates; press the reset key to reset the setting.
0xB006	Workpiece coord. computing error	Reading workpiece coordinates is in error, please press the reset key to reload.
0xB007	Servo port setting error	Station number for servo port repeats, please check the parameter setting.
0xB009	G code buffer zone error	G code buffer zone is in error, please press the reset key.
0xB00A	Invalid interpolator cmd index	Press the reset key to reload the program.
0xB00B	Interpolator cmd buffer zone access err	Interpolator command buffer zone access error. Press the reset key to reload the program.
0xB00C	Feed rate is not defined	G code error; please check G code and revise the program.
0xB00D	Invalid arc radius	G code error; please check G code and revise the program.
0xB00E	Invalid tool ID selection	G code error; 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 identical.
0xB010	Breakpoint number not found	Check if G code contains the label or line being searched; revise the program.
0xB014	Incorrect tool compen. radius	Incorrect tool compensation radius. Check G code and revise the program.
0xB015	Sync cmd error	Sync command error. Check the MLC program procedure.
0xB017	Cutter specifying error	Make sure the cutter number is within the specified range.
0xB018	Switching transition cmd prohibited	Check G code and revise the program. Check the MLC program procedure.

Alarm code	Name	Causes and troubleshooting
0xB019	Cmd error	Command error. Axis cannot be moved; please check the procedure.
0xB01A	Data full	Check G code and revise the program.
0xB01B	Spindle does not operate	The spindle does not operate during machining.
0xB020	Emergency stop	Emergency stop is pressed. Please check the connection of emergency stop
0xB021	Can't figure out chamfer/ round corner	Chamfer/ round corner cannot be figured out. Please check and verify the procedure.
0xB100	Radius compen. path interference	Please verify the path of tool compensation.
0xB101	Enable/cancel radius compen. in arc	G code error; please check G code and revise the program.
0xB102	Plane change when compensation	Changing plane is not allowed during cutter radius compensation. G code error; please check G code and revise the program.
0xB103	Radius interference	G code error; please check G code and revise the program.
0xB104	Tool compen. amount too small	G code error; please check G code and revise the program.
0xB105	Left & right compen. switch error	G code error; 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.
0xB600	Invalid G code ID	Invalid G code ID; please check G code and revise the program.
0xB603	Invalid variable symbol	Invalid variable symbol; please check G code and revise the program.
0xB604	Illegal G code symbol	Illegal G code symbol; please check G code and revise the program.
0xB605	No G code symbol	No G code symbol; please check G code and revise the program.

Alarm code	Name	Causes and troubleshooting
0xB606	Subroutine calling error	Subroutine does not exist. Please revise the program.
0xB607	Subroutine file name err	Subroutine file name is invalid; please revise the program.
0xB608	Subroutine nesting error	Number of the subroutine nest exceeds the range. Please revise the program.
0xB60A	Syntax error of G04	Verify the syntax for pause time and revise the program.
0xB60C	Arc magnification error	The syntax for arc magnification is in error, please revise the program.
0xB60D	Middle point homing err	Intermediate point for homing is not defined, please revise the program.
0xB60E	Homing when cycle mach.	Homing is not allowed during cycle machining; please revise the program.
0xB60F	G54 augmented code err	G54 augmented code error. 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.

17.4   Error message of macro configuration

Alarm code	Name	Causes and troubleshooting
0xB610	Invalid macro VAR type	Invalid macro variable type. Check the macro and revise the program.
0xB611	Macro cmd is not found	Check the macro and revise the program.
0xB612	Invalid macro cmd line No.	The label is not found when executing GO TO command. Please revise the program.
0xB613	Marco bit setup error	Check the bit setup range in macro and revise the program.
0xB614	Divided by zero in macro	Divided by zero in macro error. The denominator of the divisor cannot be zero, please revise the program.
0xB615	Macro cmd is too long	Macro command has exceeded the length, please revise the program.
0xB616	No macro cmd operand	No macro command operand. Check the macro and revise the program.
0xB617	Macro cmd error	Check the macro and revise the program.
0xB618	Macro syntax error	Macro command not found, please revise the program.
0xB619	Macro operand syntax err	Macro operand syntax error. Syntax of the operand in macro is not complete. Please check the macro and revise the program.
0xB61A	Illegal macro cmd	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
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.
0xB630	Excess following error	Check the servo connection. Make sure the setting of servo parameter is correct.
0xB631	Hardware limit error	Revise the program. Check if the limit switch is valid or polarity setting is correct.

Alarm code	Name	Causes and troubleshooting
0xB632	The 1st software limit error	Revise the program. Check the parameter setting of the first software limit
0xB634	The 2nd software limit error	Revise the program. Check the parameter setting of the second software limit.
0xB636	Origin sensor error	Make sure the installation and the polarity settings of the origin switch are correct. Ensure the parameter setting of the distance for looking the origin is correct.
0xB640	Temperature sensor err 1	Temperature sensor error 1. Make sure the power supply of the module is normal; Make sure each connector is properly connected
0xB641	Temperature sensor err 2	Temperature sensor error 2. Make sure the power supply of the module is normal; Make sure each connector is properly connected
0xB642	Temperature sensor err 3	Temperature sensor error 3. Make sure the power supply of the module is normal; Make sure each connector is properly connected
0xB643	Temperature sensor err 4	Temperature sensor error 4. Make sure the power supply of the module is normal; Make sure each connector is properly connected.

17.5   HMI alarms (Code: 0x3010 ~ 0x3FFF)

Description of HMI interface error messages

Alarm code	Name	Causes and troubleshooting
0x3010	HMI com. interface error	Restart the controller or send it back for servicing.
0x3011	HMI com. memory zone error	Restart the controller or send it back for servicing.
0x3012	HMI interface cmd zone error	Restart the controller or send it back for servicing.
0x3013	HMI interface memory error	Restart the controller or send it back for servicing.
0x3014	HMI interface com. port error	Restart the controller or send it back for servicing.
0x3015	MLC interface memory error	Restart the controller or send it back for servicing.
0x3016	HMI file transmission error	Restart the controller or send it back for servicing.
0x3017	HMI data transmission error	Restart the controller or send it back for servicing.
0x3100	Illegal file name	Check if the file exists; rename the file.
0x3101	Subroutine nests called overrange	Reduce the subroutine nests being called.
0x3102	Non-G code character contained	Check G code and revise the program. Delete the symbol that is not allowed in G code.
0x3103	Memory diagnosis error	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.
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.

Alarm code	Name	Causes and troubleshooting
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	Please check all the parameter values of the system and make sure they are within the range.
0x3207	Function library loading error	An error has occurred when loading the function library for connecting with the external device via RS-485. Please make sure the setting of Base Port in ScreenEditor is correct and reload the program of software interface.
0x3208	Machine to be locked	Expiry date is close; the machine will be locked when expired and machining cannot be done. 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.

17.6   Servo drive alarms

Display	Name	Description
AL001	Overcurrent	The current of the main circuit is 1.5 times more than the instantaneous current of the motor.
AL002	Overvoltage	The voltage of the main circuit is higher than the standard voltage.
AL003	Undervoltage	The voltage of the main circuit is lower than the standard voltage.
AL004	Motor combination error	The drive corresponds to the wrong motor.
AL005	Regeneration error	Regeneration control is in error.
AL006	Overload	The motor and the drive is overload.
AL007	Overspeed	The motor speed exceeds the normal speed.

Display	Name	Description
AL008	Abnormal pulse command	The input frequency of the pulse command is over the allowable value of the hardware interface.
AL009	Excessive deviation of position command	The deviation of position command exceeds the allowable setting value.
AL010	Reserved	Reserved
AL011	Encoder error	The encoder produces abnormal pulse.
AL012	Adjustment error	When executing electrical adjustment, the adjusted value exceeds the allowable value.
AL013	Emergency stop	Emergency stop switch is on.
AL014	Reverse limit error	Reverse limit switch is on.
AL015	Forward limit error	Forward limit switch is on.
AL030	Motor crash error	The motor crashes the equipment, reaching the torque setting of P1-57 and exceeds the time set by P1-58.
AL031	Incorrect wiring of the motor power line U, V, W, GND	Incorrect wiring of the motor power line U, V, W, GND.
AL040	Excessive deviation of full closed-loop position control	Excessive deviation of full closed-loop position control.
AL099	DSP firmware upgrade	EEPROM has not been reset after upgrading the firmware. The fault can be cleared when firstly set P2-08 to 30. Then set P2-08 to 28. And re-power on the drive.
AL185	Abnormal CAN Bus hardware	The communication of CAN Bus is broken down or Error Rx/Tx Counter is over 128.
AL302	The synchronized signal of CANopen is sent too fast	The synchronized signal, SYNC of CANopen is sent too fast.
AL303	The synchronized signal of CANopen is sent too slow	The synchronized signal, SYNC of CANopen has not been received in time.
AL304	CANopen IP command is failed	Command cannot be issued in CANopen IP mode.