

GFZ-61397E B-61397E

GE Fanuc CNC

0/00/0-Mate Operation and Maintenance Handbook



Presented By: CNC Center

For Product Needs Please Visit:

<http://www.cnccenter.com/>

OR Email:

sales@cnccenter.com

OR Call:

1-800-963-3513



GE Fanuc Automation

Computer Numerical Control Products

Series 0 / 00 / 0-Mate

Operation and Maintenance Manual

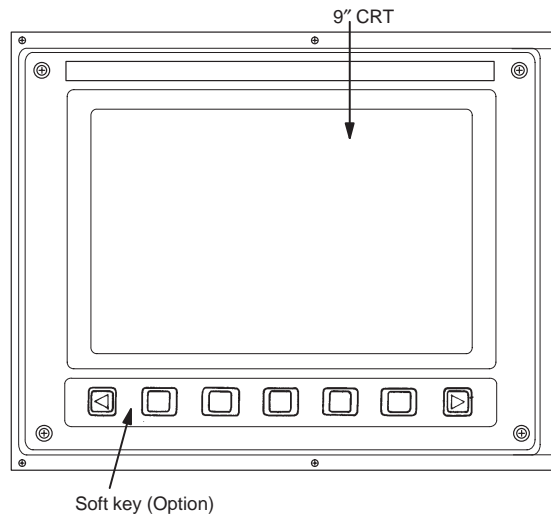
GFZ-61397E/02

June 1996

1. CRT/MDI PANEL

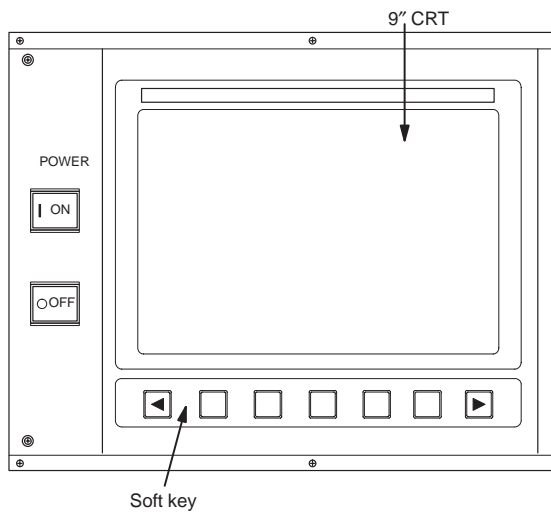
1.1 Standard CRT/MDI Panel

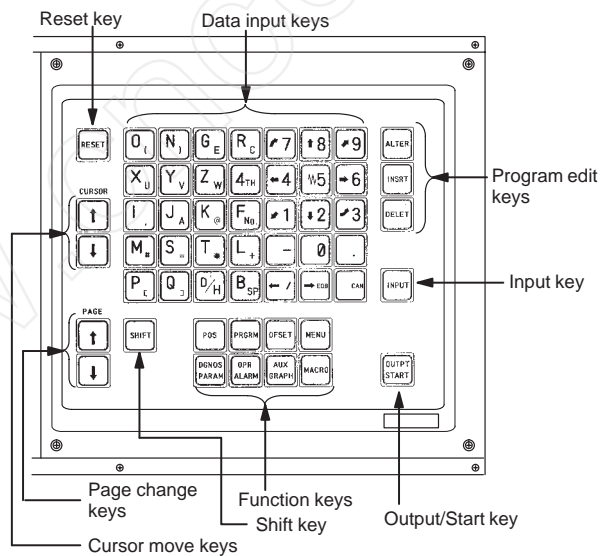
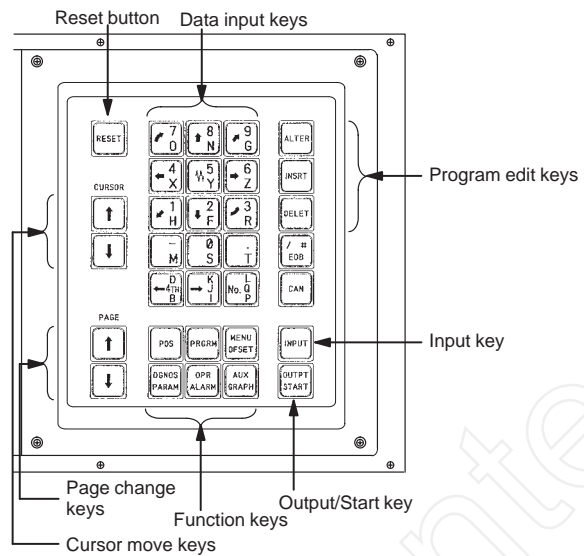
M series



1.2 Full Key CRT/MDI Panel

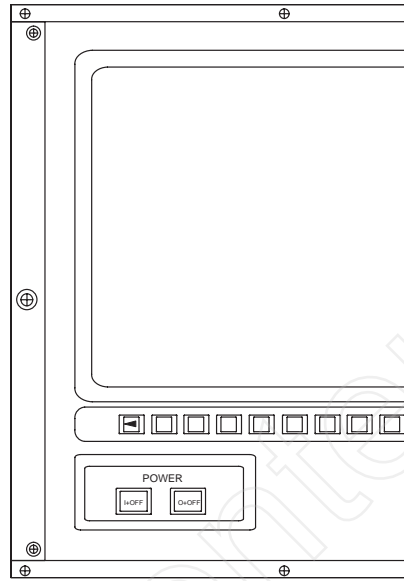
M series





1. CRT/MDI PANEL

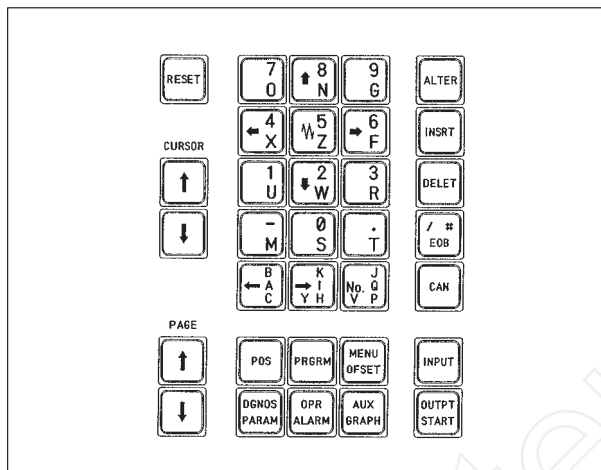
1.3 CRT/MDI with MMC



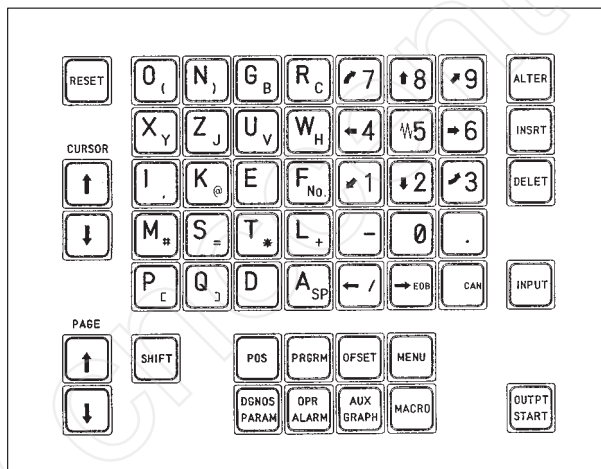
1. CRT/MDI PANEL

1.4 MDI Keyboard

- Standard MDI keyboards (T series)

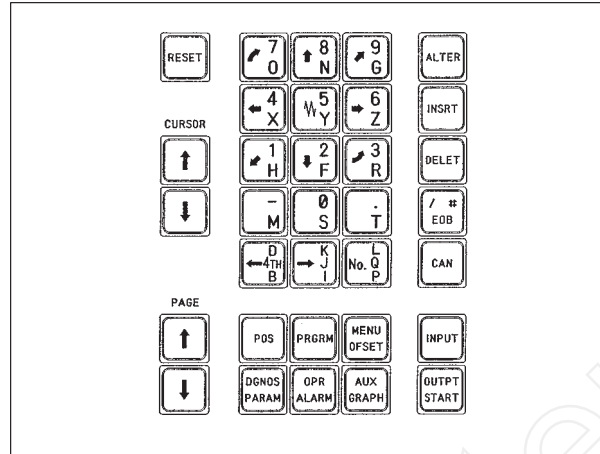


- Full MDI keyboards (T series)

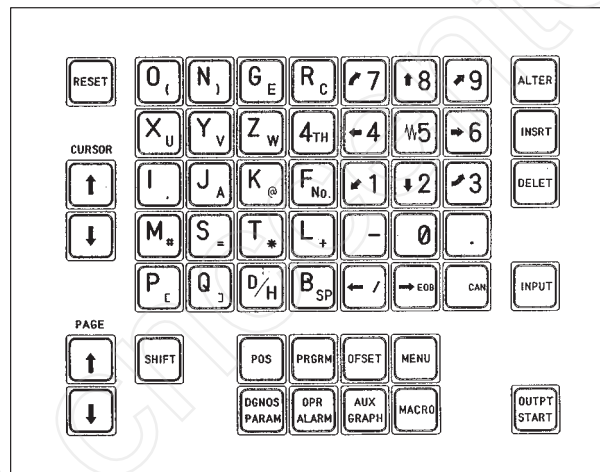


- Standard MDI keyboards (M series)

1

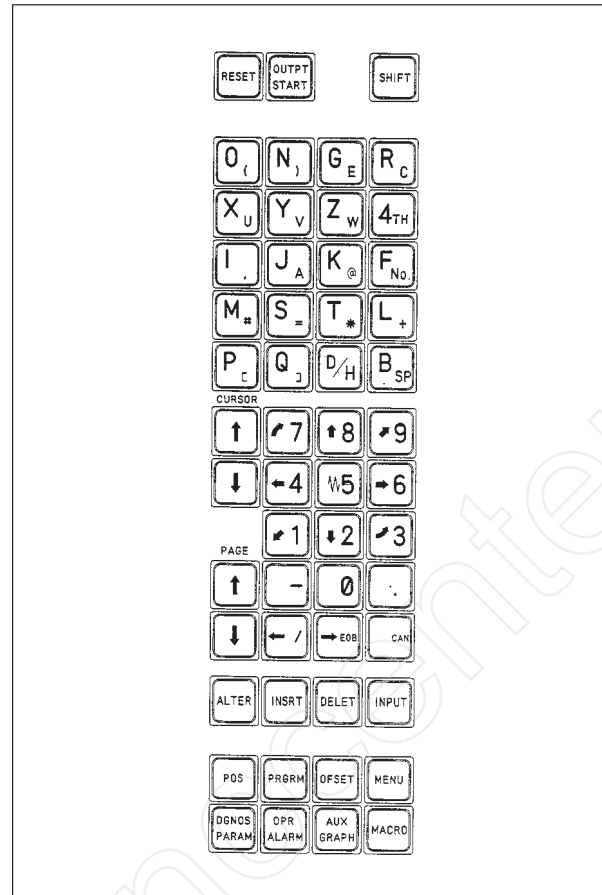


- Full MDI keyboards (M series)

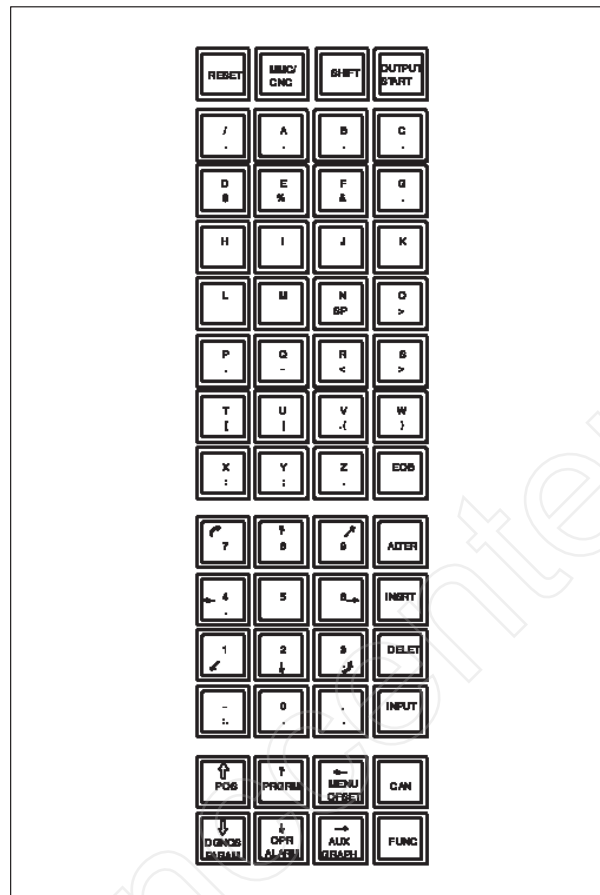


1. CRT/MDI PANEL

- Full MDI keyboards (14" CRT for M series)














- MDI keyboards with MMC





















1

1. CRT/MDI PANEL

1.5 Explanation of the Keyboard

Number	Name	Explanation
1	Power ON and OFF buttons  	Press these buttons to turn CNC power ON and OFF.
2	RESET key 	Press this key to reset the CNC, to cancel an alarm, etc.
3	START key 	This key is used to start MDI operation or automatic operation, depending on the machine. Refer to the manual provided by the machine tool builder. This key is also used to output data to an input/output device.
4	Soft keys (option)	The soft keys have various functions, according to the applications. The soft key functions are displayed at the bottom of the CRT screen. Soft key of left edge  : Return menu key Soft key of right edge  : Continuous menu key
5	Address and numeric keys   4 ...	Press these keys to input alphabetic, numeric, and other characters.
6	SHIFT key  (Full MDI keyboard)	Some keys have two characters on their keytop. Pressing the <SHIFT> key switches the characters. Special character ^ is displayed on the screen when a character indicated at the bottom right corner on the keytop can be entered.
7	INPUT key 	When an address or a numerical key is pressed, the data is input to the buffer, and it is displayed on the CRT screen. To copy the data in the key input buffer to the offset register, etc., press the  key. This key is also used to input data from an input/output device.

Number	Name	Explanation
8	Cancel key 	Press this key to delete the input data or the last character in the key input buffer.
9	Program edit keys   	Press these keys when editing the program.  : Alteration  : Insertion  : Deletion
10	Function keys   ...	Press these keys to switch display screens for each function.
11	Cursor move keys CURSOR  	There are two different cursor move keys.  : This key is used to move the cursor in an upward or reverse direction.  : This key is used to move the cursor in a downward or forward direction.
12	Page change keys PAGE  	Two kinds of page change keys are available.  : This key is used to changeover the page on the CRT screen in the reverse direction.  : This key is used to changeover the page on the CRT screen in the forward direction.
13	MMC/CNC change key 	Selects whether the MMC screen or CNC screen is displayed on the CRT.

1

1. CRT/MDI PANEL

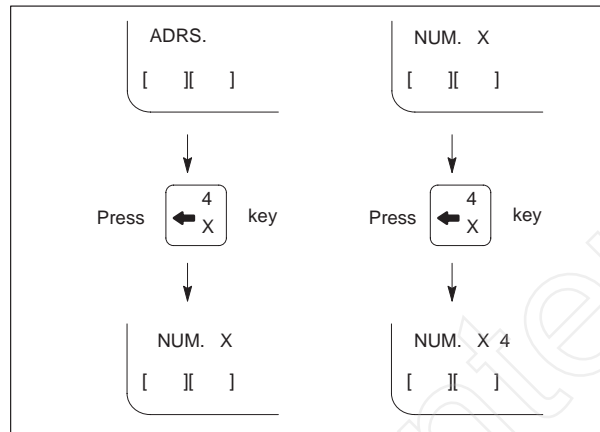
1.6 Key Input

- **For standard MDI keyboard**

On the standard MDI keyboard, the same key is used to input both an address and a numeric value.

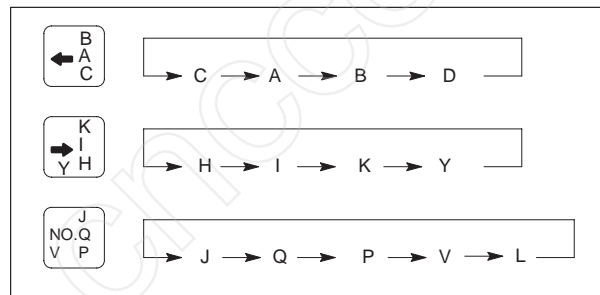
When "ADRS." is displayed on the top of the key input buffer, addresses can be input.

When "NUM." is displayed on the top of the key input buffer, numeric values can be input.

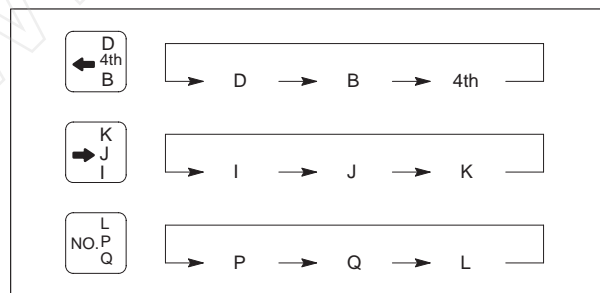


The following keys are used for inputting multiple addresses. They may be not displayed on the screen depending on the options used.

T series



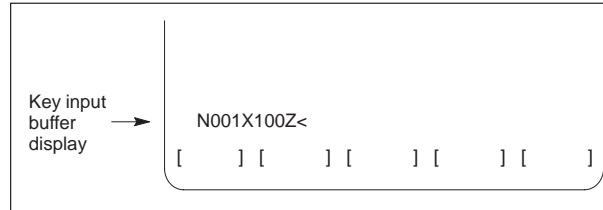
M series



- **For full MDI keyboard**

A "<" is displayed at the end of the key input buffer indicating the input position of the next character.

1



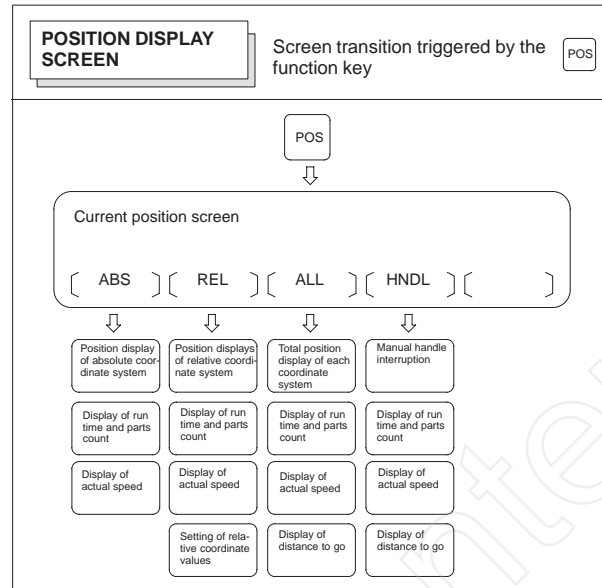
Key input buffer display

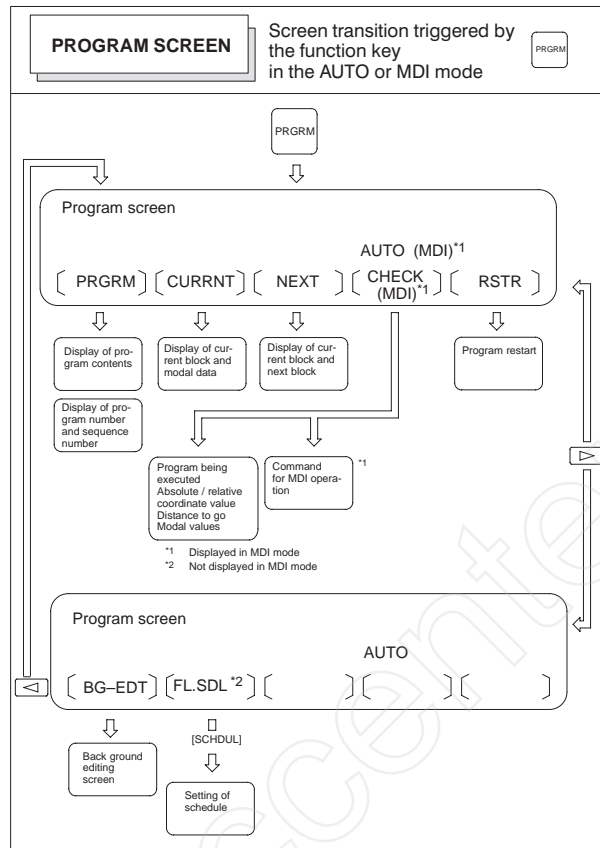
To input the symbol indicated at the lower part of a key top, press the SHIFT

key to change the prompt < to \wedge . Then press the key.

2. CRT/MDI OPERATION

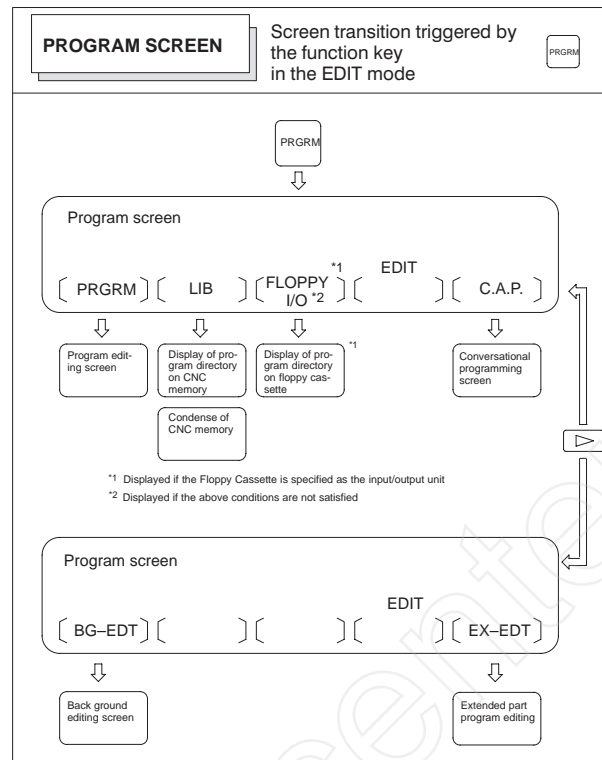
2.1 Screen Transition Triggered by the Each Function Key

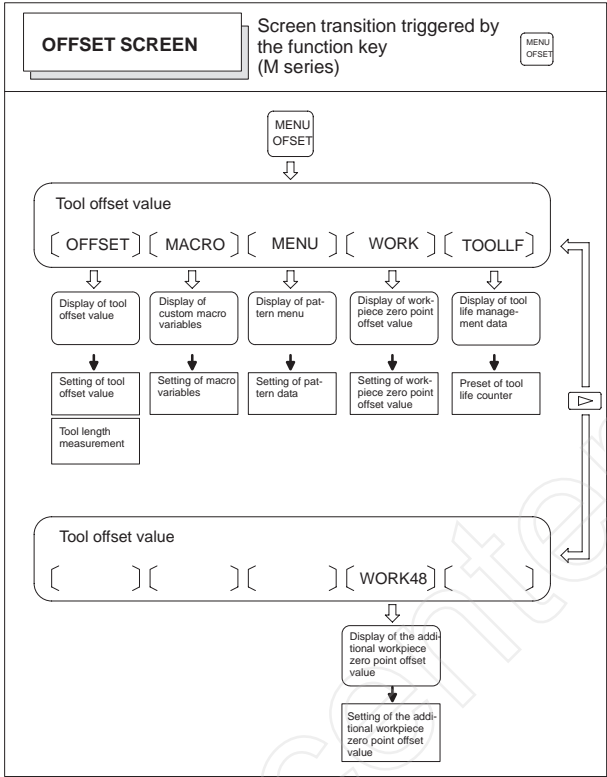
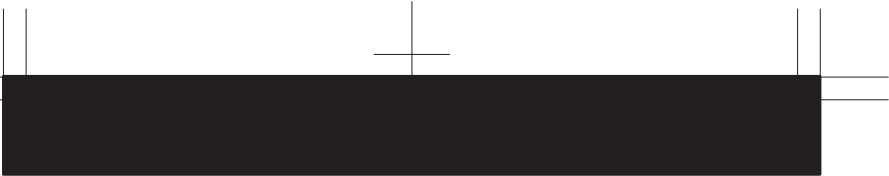




2

2. CRT/MDI OPERATION



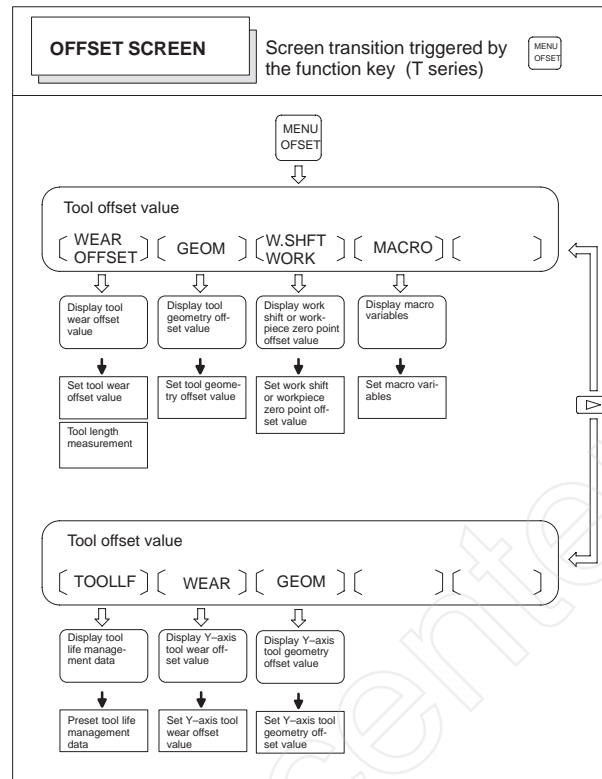


2

www.cnccookbook.com



2. CRT/MDI OPERATION



PARAMETER/DIAGNOSTIC SCREEN

Screen transition triggered by the function key

DGNOS
PARAM

DGNOS
PARAM

Parameter screen

[PARAM] [DGNOS] [] [SV-PRM*] []

Display of pa-
rameter
screen

Setting of param-
eter

Setting of pitch
error compensa-
tion data

Display of
setting data

Setting of set-
ting data

Setting of se-
quence number
comparison and
stop

Setting of parts
count

Display of run
time, parts count

Display of date
and clock

Setting of date
and clock

Display of
diagnosis
screen

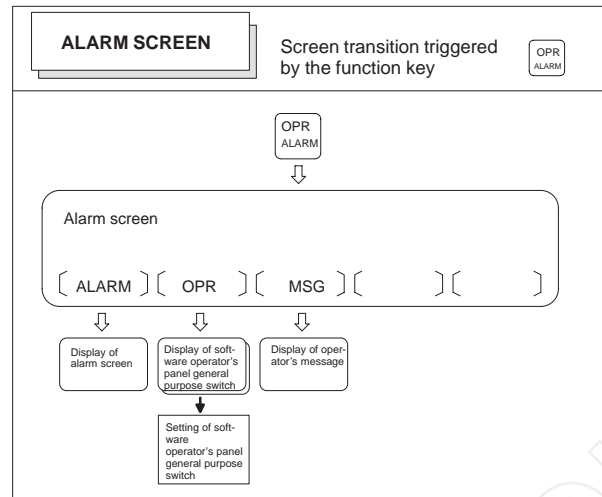
Display of
servo setting
screen

Display of
servo tuning
screen

* The servo setting/tuning screen can be suppressed if bit 0 of parameter 0389 is specified accordingly.


2


2. CRT/MDI OPERATION



2.2 CRT/MDI Operation

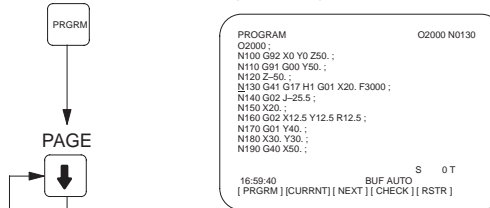
Command display

- Function button 

During the AUTO mode, press  (page) or the corresponding soft key to cause any of the following four types of displays to appear.

2

- (1) A program being currently executed is displayed.



```

PROGRAM                                Q2000 N0130
O2000:
N100 G82 X0 Y0 Z50.;
N110 G91 G00 Y60.;
N120 Z-50.;
N130 G41 G17 H1 G01 X20. F3000;
N140 G02 J-25.5;
N150 X20.;
N160 G02 X12.5 Y12.5 R12.5;
N170 G01 Y40.;
N180 X30. Y30.;
N190 G40 X50.;

16:59:40          BUF AUTO      S 0 T
[PRGRM][CURRNT][NEXT][CHECK][RSTR]
  
```

The cursor is set to the beginning of the program being executed.

- (2) The command currently being executed and the modal values specified before are displayed.

```

PROGRAM                                Q2000 N0130
(CURRNT)                                (MODAL)
X 20.000 G67 G01 F 3000
G01 F 3000 G54 G17 R
G17 H 1 G64 G91 P
G69 G22 Q
G15 G94 H
G25 G21 M
G41 G41 S
G40 T
G80 G80 S
G50 S 0 T0000
02:50:52          BUF AUTO
[PRGRM][CURRNT][NEXT][CHECK][RSTR]
  
```

- (3) The command currently being executed and the next command to be executed are displayed.

```

PROGRAM                                Q2000 N0130
(CURRNT)                                (NEXT)
X 20.000 J -25.500
G01 F 3000 G02
G17 H 1
G41
G80 G80
02:52:14          BUF AUTO      S 0 T
[PRGRM][CURRNT][NEXT][CHECK][RSTR]
  
```

- (4) Program check

The block currently being executed, the current position, and the modal values specified before are displayed.

```


PROGRAM CHECK                            Q2000 N0130
N130 G41 G17 H1 G01 X20. F3000;
N140 G02 J-25.5;
N150 X20.0;
N160 G02 X12.5 Y12.5 R12.5;
(RELATIVE) (DIST TO GO) (G)
X 17.600 X 2.400 G01 G21 G50
Y 50.000 Y 0.000 G17 G41 G67
Z 0.000 Z 0.000 G91 G49 G54
G22 G80 G54
G94 G98 G69
F 3000 P H 1 S
R Q M T
ACT F 3000 MMM S 0 T
02:53:16          BUF AUTO
[PRGRM][CURRNT][NEXT][CHECK][RSTR]
  
```

* Besides the above displays, the program restart screen is displayed in some cases. (Option)

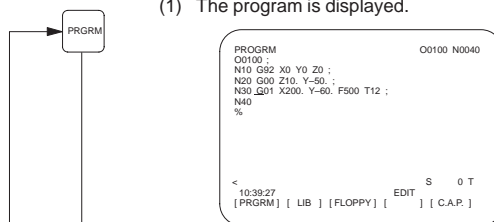
2. CRT/MDI OPERATION

Program display

- Function button 

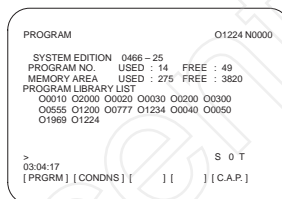
During the EDIT mode, press the  key several times or the corresponding soft key to cause either of the following two types of displays to appear.

- (1) The program is displayed.



The program and sequence numbers are displayed at the upper right section of the screen.


- (2) The amount of program memory in use is displayed.




- Number of registered programs
- Memory area in use
- Program list



* If the directory display of floppy cassette is provided, a file list, in addition to the above displays, appears on the screen.

Reset


Press the . This key is usually used to reset the alarm.


When the  is pressed, the NC enters one of the states listed below.

Before a reset		After a reset
A tool movement command continues to be executed.		The tool decelerates and stops. The unexecuted amount of movement disappears.
An M, S, or T continues to be sent out.		A send-out sequence is terminated. Refer to the machine tool builder's manual for what occurs on the machine side.
When the buffer is loaded with one block	MDI mode	The contents of the buffer are not erased.
	Other modes	The contents of the buffer are erased. The BUF display disappears.

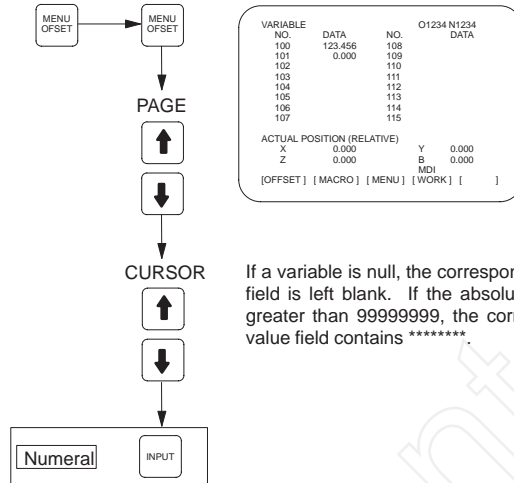
In any case, pressing the  resets the NC. In a mode other than the MDI mode,  causes the labels to be skipped.

Custom macro variable display/setting

- Function button 

Press the  key several times or the corresponding soft key to select the desired screen.

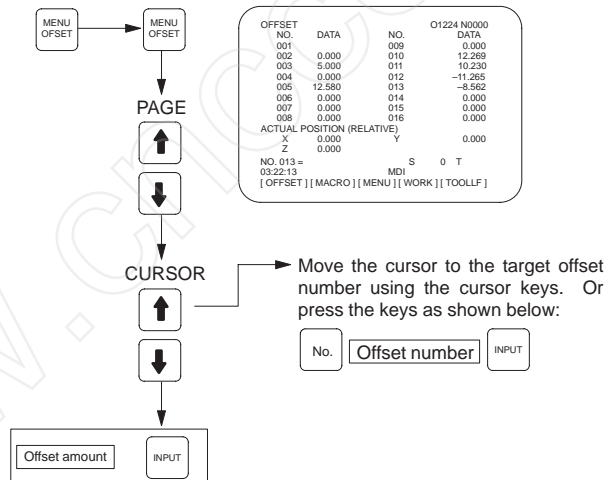
2



VARIABLE		O1234 N1234	
NO.	DATA	NO.	DATA
100	123.456	108	
101	0.000	109	
102		110	
103		111	
104		112	
105		113	
106		114	
107		115	
ACTUAL POSITION (RELATIVE)			
X	0.000	Y	0.000
Z	0.000	B	0.000
[OFFSET] [MACRO] [MENU] [WORK] []			

If a variable is null, the corresponding value field is left blank. If the absolute value is greater than 99999999, the corresponding value field contains *****.

Tool compensation value display/setting



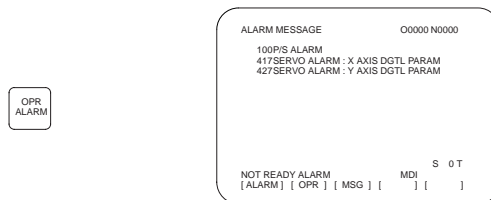
OFFSET		O1224 N0000	
NO.	DATA	NO.	DATA
001		009	0.000
002	0.000	010	12.289
003	5.000	011	10.230
004	0.000	012	-11.265
005	12.580	013	-8.562
006	0.000	014	0.000
007	0.000	015	0.000
008	0.000	016	0.000
ACTUAL POSITION (RELATIVE)			
X	0.000	Y	0.000
Z	0.000	S	0 T
NO. 013 = MDI			
03.22.13 [OFFSET] [MACRO] [MENU] [WORK] [TOOLFF]			

Move the cursor to the target offset number using the cursor keys. Or press the keys as shown below:


No. INPUT

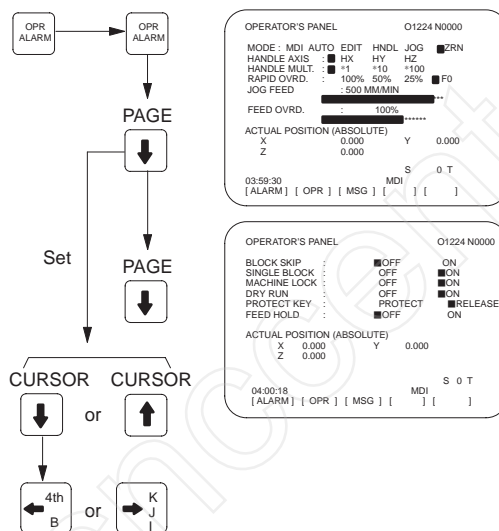
2. CRT/MDI OPERATION

Alarm display



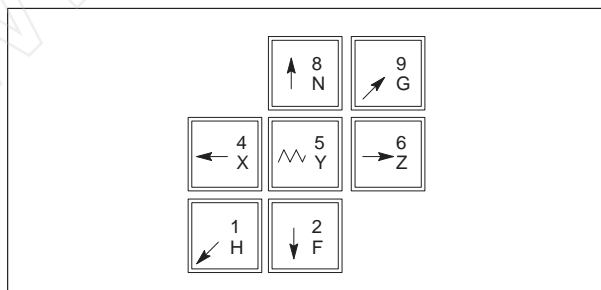
Software operator's panel display/setting

Press the  key several times or the corresponding soft key to select the target screen.



The switches and controls on the CRT/MDI panel can be used in place of the counterparts on the machine operator's panel. (FANUC MPC is required.)

- Use the buttons shown below for jog feed operations.



2.3 CRT/MDI Operation and Display (M Series)

Current position display

- Function button POS

Press ↓ (page) or the corresponding soft key to cause any of the following three types of displays to appear.

2

(1) Absolute coordinates

Distance from the programmed zero point

POS

↓

↓

ACTUAL POSITION (ABSOLUTE)	
O0010	N0000
X	123.456
Y	363.233
Z	0.000
RUN TIME	0H 1M
ACT F	3000 M/M
01:35:22	
PART COUNT	1
CYCLE TIME	0H 1M33S
S	0 T
BUF AUTO	
[ABS]	[REL] [ALL] [HNDL] []

(2) Relative coordinates

Any tool position can be set to 0 by X ⇔

CAN. This holds true also of Y and Z.

ACTUAL POSITION (RELATIVE)	
O0010	N0000
X	123.456
Y	363.233
Z	0.000
RUN TIME	0H 1M
ACT F	3000 M/M
01:36:12	
PART COUNT	1
CYCLE TIME	0H 1M33S
S	0 T
BUF AUTO	
[ABS]	[REL] [ALL] [HNDL] []

(3) General coordinates

Absolute coordinates
Relative coordinates
Machine coordinates

ACTUAL POSITION		O1000 N0010	
(RELATIVE)	(ABSOLUTE)		
X 246.912	X 123.456		
Y 913.780	Y 456.890		
Z 1578.246	Z 789.123		
(MACHINE)	(DISTANCE TO GO)		
X 0.000	X 0.000		
Y 0.000	Y 0.000		
Z 0.000	Z 0.000		
RUN TIME	0H 4M	PART COUNT	1
ACT F	0 M/M	CYCLE TIME	0H 1M38S
01:54:57		S	0 T
		MDI	
[ABS]	[REL] [ALL] [HNDL] []		

The target screen can be selected directly by the corresponding soft key.

The operating time and parts count are displayed.


Two types of operating time and parts count are displayed on the coordinate display screen. (position)

2. CRT/MDI OPERATION

Setting data display/setting

- Function button 

(Parameter pages 1 and 2 are the setting data screens.)

When the title parameter is not on the screen, press the  key or the corresponding soft key to select the parameter screen.

PARAMETER O1224 N0000

(SETTING 1)

REVX = 0
REYV = 0
TVON = 0
ISO = 0 (0:EIA 1:ISO)
INCH = 0 (0:MM 1:INCH)
IO = 0
ABS = 0 (0:INC 1:ABS)
SEQ = 0

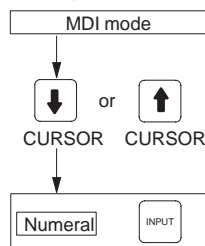
CLOCK 10/01/10 03:30:18

NO. REVX = 03:30:09 MDI S 0 T
[PARAM] [DGNOS] [] [SV-PRM] []



(page)

[Setting]



PARAMETER O1224 N0000

(SETTING 2)

PWE = 1 (0:DISABLE 1:ENABLE)
REV4 = 0
TAPEF = 0
(SEQUENCE STOP)
PRGNO = 0
SEQNO = 0

PART TOTAL = 17
PART REQUIRED = 50
PART COUNT = 17
RUN TIME OH 4M CYCLE TIME OH OM 2S

NO. PWE = 03:35:07 MDI S 0 T
[PARAM] [DGNOS] [] [SV-PRM] []

Display Item	0	1
REVX	X-axis mirror image OFF	X-axis mirror image ON
REYV	Y-axis mirror image OFF	Y-axis mirror image ON
TVON	No TV check is made.	A TV check is made.
ISO	EIA code output (during punch)	ISO code output (during punch)
INCH	Metric input	Inch input
I/O (Note)	Reader/punch interface channel is selected.	
ABS	Incremental command (MDI mode)	Absolute command (MDI mode)
SEQ	Sequence numbers are not inserted automatically when a program is entered from the MDI.	Sequence numbers are inserted automatically when a program is entered from the MDI.
PWE	Parameter writing is disabled.	Parameter writing is enabled.
REV4	Fourth-axis mirror image OFF	Fourth-axis mirror image ON
TAPEF	F10/11 table format is not used	F10/11 table format is used

NOTE See Page 33 for details.

2.4 CRT/MDI Operation and Display (T Series)

Current position display

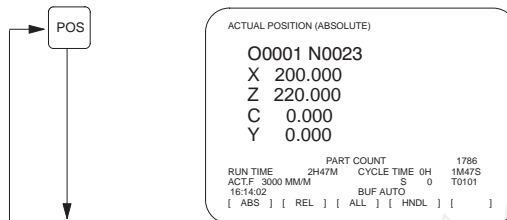
- Function button POS

Press ↓ (page) or the corresponding soft key to cause any of the following three types of displays to appear.

2

(1) Absolute coordinates

Distance from the programmed zero point

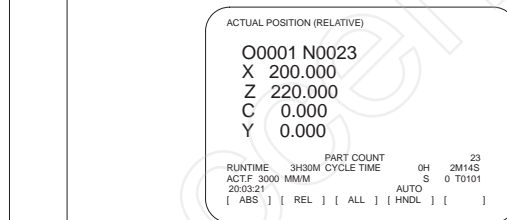


ACTUAL POSITION (ABSOLUTE)			
O0001 N0023			
X	200.000		
Z	220.000		
C	0.000		
Y	0.000		
RUN TIME		PART COUNT	1786
2H47M		CYCLE TIME	0H 1M47S
ACT.F 3000 MM/M		S	0 T0101
16:14:02		BUF AUTO	
[ABS]	[REL]	[ALL]	[HNDL]

(2) Relative coordinates

Any tool position can be set to 0 by X ⇒

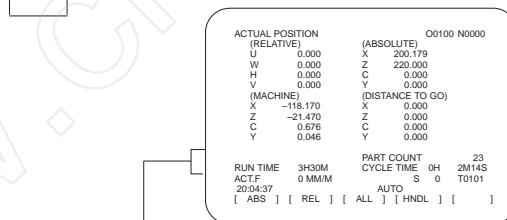
CAN. This holds true also of Z and Y.



ACTUAL POSITION (RELATIVE)			
O0001 N0023			
X	200.000		
Z	220.000		
C	0.000		
Y	0.000		
RUNTIME		PART COUNT	23
3H30M		CYCLE TIME	0H 2M14S
ACT.F 3000 MM/M		S	0 T0101
20:03:21		AUTO	
[ABS]	[REL]	[ALL]	[HNDL]

(3) General coordinates

Absolute coordinates
Relative coordinates
Machine coordinates



ACTUAL POSITION (RELATIVE)		(ABSOLUTE)		O0100 N0000	
U	0.000	X	200.179		
W	0.000	Z	220.000		
H	0.000	C	0.000		
V	0.000	Y	0.000		
(MACHINE)		(DISTANCE TO GO)			
X	-119.170	X	0.000		
Z	-21.470	Z	0.000		
C	0.676	C	0.000		
Y	0.046	Y	0.000		
RUN TIME		PART COUNT	23		
3H30M		CYCLE TIME	0H 2M14S		
ACT.F 0 MM/M		S	0 T0101		
20:04:37		AUTO			
[ABS]	[REL]	[ALL]	[HNDL]		

The target screen can be selected directly by the corresponding soft key.


The operating time and part count are displayed. Two types of operating time and parts count are displayed on the coordinate display screen (position).

2. CRT/MDI OPERATION

Setting data display/setting

- Function button 

(Parameter pages 1 and 2 are the setting data screens.)

When the title "parameter" is not on the screen, press the  key or the corresponding soft key to select the parameter screen.

PARAMETER O0001 N0001

(SETTING 1)

TVON = 1

ISO = 1 (0:EIA 1:ISO)

INCH = 0 (0:MM 1:INCH)

I/O = 0

SEQ = 1

CLOCK 94/05/16 12:11:52

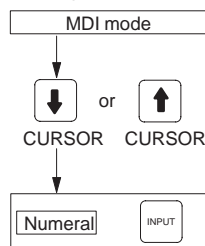
NO. TVON 21:35:25 MDI S 0 T0101

[PARAM] [DGNOS] [] [SV-PRM] []



(page)

[Setting]



PARAMETER O0001 N0001

(SETTING 2)

PWE = 0 (0:DISABLE 1:ENABLE)

TAPEF = 0

(SEQUENCE STOP)

PRGNO = 0

SEQNO = 0

PART TOTAL = 23

PART REQUIRED = 0

PART COUNT = 23

RUN TIME 3H36M CYCLE TOME 0H 0M 0S

NO. PWE 21:35:25 MDI S 0 T0101

[PARAM] [DGNOS] [] [SV-PRM] []


Display Item	0	1
TVON	No TV check is made.	A TV check is made.
ISO	EIA code output (during punch)	ISO code output (during punch)
INCH	Metric input	Inch input
I/O (Note)	Reader/punch interface channel is selected.	
SEQ	Sequence numbers are not inserted automatically when a program is entered from the MDI.	Sequence numbers are inserted automatically when a program is entered from the MDI.
PWE	Parameter writing is disabled.	Parameter writing is enabled.
TAPEF	F10/11 table format is not used.	F10/11 table format is used.

NOTE See Page 33 for details.

2. CRT/MDI OPERATION

2.6 Data Input/Output (FANUC Cassette)

Setting the beginning of the file


- 1 Select the EDIT mode
- 2 Press the  key several times to select the program list screen.

```
PROGRAM                                O1224 N0000
SYSTEM EDITION  0466 - 25
PROGRAM NO.     USED : 14  FREE : 49
MEMORY AREA    USED : 275 FREE : 3820
PROGRAM LIBRARY LIST
O0010 O2000 O0020 O0030 O0200 O0300
O0555 O1200 O0777 O1234 O0040 O0050
O1989 O1224

>                                S 0 T
03:04:17
[PRGRM] [CONDNS] [ ] [ ] [C.A.P.]
```

- 3 Key in address N.
- 4 Key in the target file number.
 - N0 ⇒ Locates the beginning of the cassette.
This is used regardless of whether the file is on the floppy disk.
 - N1 ⇒ Locates the beginning of the cassette.
This is used when the file is on the floppy disk.
 - N2 to N9999 ⇒ Locates the beginning of any file.

CNC parameter output

- 1 Select the EDIT mode
- 2 Press the  key several times to select the parameter screen.



```
PARAMETER                                O1224 N0000
(SETTING 1)
REVX = 0
REY = 0
TVON = 0
ISO = 0 (0:EIA 1:ISO)
INCH = 0 (0:MM 1:INCH)
I/O = 0
ABS = 0 (0:INC 1:ABS)
SEQ = 0

CLOCK 10/01/10
03:30:18
NO. REVX = S 0 T
03:30:09
[PARAM] [DGNOS] [ ] [SV-PRM] [ ]
```


- 3 Press the  key, and output begins.

NOTE Parameter Nos. nine hundreds (900 to 999) are not output.

PMC parameter output

- 1 Select the EDIT mode.
- 2 Press the  key several times to select the DGNOS (diagnose) screen.
- 3 Press the  key, and output begins.

Program output

- 1 Select the EDIT mode.
- 2 Press the  key several times to select the program list screen.


PROGRAM O1224 N0000

SYSTEM EDITION 0486 - 25
 PROGRAM NO. USED : 14 FREE : 49
 MEMORY AREA USED : 275 FREE : 3820

PROGRAM LIBRARY LIST
 O0010 O0030 O0030 O0030 O0200 O0300
 O0555 O1200 O0777 O1234 O0040 O0050
 O1969 O1224

> S O T
 03:04:17
 [PRGRM] [CONDNS] [] [] [C.A.P.]


2

- 3 Key in address O.
- 4 Key in the target program number.
- 5 Press the  key, and output begins.

* All-program output: 0-9999



Offset output

- 1 Select the EDIT mode.
- 2 Press the  key several times to select the offset screen.

OFFSET O1224 N0000

NO.	DATA	NO.	DATA
001	0.000	009	0.000
002	0.000	010	12.269
003	5.000	011	10.230
004	0.000	012	-11.265
005	12.580	013	-8.562
006	0.000	014	0.000
007	0.000	015	0.000
008	0.000	016	0.000



ACTUAL POSITION (RELATIVE)
 X 0.000 Y 0.000
 Z 0.000

NO. 013 = S O T
 03:22:13 MDI
 [OFFSET] [MACRO] [MENU] [WORK] [TOOLFF]

- 3 Press the  key, and output begins.

Conversational mode data output

[M series]

- 1 Select the EDIT mode.
- 2 Press the  key several times to select the conversational mode data screen.
- 3 Press the  key, and output begins.


2. CRT/MDI OPERATION

CNC parameter input

- 1 Set setting data PWE to 1 (page 2 of the parameter screen).

This is set in the MDI mode or at an emergency stop.

PARAMETER		O1224 N0000
(SETTING 2)		
PWE	=	1 (0:DISABLE 1:ENABLE)
REV4	=	0
TAPEF	=	0
(SEQUENCE STOP)		
PRGNO	=	0
SEQNO	=	0
PART TOTAL	=	17
PART REQUIRED	=	50
PART COUNT	=	17
RUN TIME	=	OH 4M CYCLE TIME OH OM 2S
NO. PWE	=	S 0 T
03:35:07		MDI
[PARAM]	[DGNOS]	[] [SV-PRM] []



NOTE Alarm P/S100 occurs at this point. Press the  key again to cause the parameter screen to appear.

- 2 Select the EDIT mode.

* Release the emergency stop condition.



- 3 Press the  key, and input begins.

* Alarm P/S000 occurs at this point. Turn the CNC power off and on again.

* If you want to enter parameters during the emergency stop state, press and hold down the  Key, then press the  Key. In this case, it is not necessary to select the EDIT mode.


NOTE It is impossible to enter parameter Nos. nine hundreds (900 to 999).

PMC parameter input

- 1 Select the EDIT mode.
- 2 Locate the beginning of the file.
- 3 Disable program protection (KEY=1).
- 4 Turn to 1 at setting parameter PWE.
- 5 Press the  key several times to select the DGNOS (diagnose) screen.
- 6 Press the  key, and input begins.

NOTE PWE should be 1 in case of parameter 393#7=0.

Program input

- 1 Select the EDIT mode.
- 2 Locate the beginning of the file.
- 3 Disable program protection (KEY=1).
- 4 Press the  key several times to select the program screen.

PROGRAM		O1224 N0000
SYSTEM EDITION 0466 - 25		
PROGRAM NO.	USED : 14	FREE : 49
MEMORY AREA	USED : 275	FREE : 3820
PROGRAM LIBRARY LIST		
O0010 O2000 O0020 O0030 O0200 O0300		
O0555 O1200 O0777 O1234 O0040 O0050		
O1969 O1224		
S 0 T		
> 03:04:17		
[PRGRM]	[CONDNS]	[] [C.A.P.]

5 Press the key, and input begins.

* When only one program is entered.

6 Key in address O.

7 Key in the target program number.

8 Press the key, and input begins.

2

Offset input

1 Select the EDIT mode.

2 Locate the beginning of the file.

3 Disable program protection (KEY=1).

4 Press the key several times to select the offset screen.

OFFSET NO.	DATA	NO.	O1224 N0000 DATA
001		009	0.000
002	0.000	010	12.269
003	5.000	011	10.230
004	0.000	012	-11.265
005	12.580	013	-8.562
006	0.000	014	0.000
007	0.000	015	0.000
008	0.000	016	0.000
ACTUAL POSITION (RELATIVE)			
X	0.000	Y	0.000
Z	0.000		
NO. 013=			
03:22:13			
[OFFSET] [MACRO] [MENU] [WORK] [TOOLLF]			

5 Press the key, and input begins.

Conversational mode data input

[M series]

1 Select the EDIT mode.

2 Locate the beginning of the file.

3 Disable program protection (KEY=1).

4 Press the key several times to select the program list screen.

5 Key in address 0.

6 Key in any program number.

7 Press the key, and input begins.

8 Select the AUTO mode.

9 A program entered before is executed.

* Be cautious about the following parameter.

PRM, No. 015	Caution	*	*	*	*	*	*	*
--------------	---------	---	---	---	---	---	---	---

CAUTION

When a decimal point is omitted from an address in which it can be used:

1 : mm, inch, and s units (usually)

0 : Least input increment (at data input time)

2. CRT/MDI OPERATION

Parameters related to data input/output

To use the FANUC floppy cassette, set the parameters as follows:

Setting: I/O = 0 (Note)

Parameter: ISO = 1

PRM, No. 002	1	*	*	*	*	0	*	1
--------------	---	---	---	---	---	---	---	---

PRM, No. 552	10	(4800BPS)
--------------	----	-----------

PRM, No. 010	*	*	*	Note	*	*	*	*
--------------	---	---	---	------	---	---	---	---

NOTE1 1 : Protects program numbers 9000s (9000 to 9999).
0 : Enables editing of program numbers 9000s (9000 to 9999).

PRM, No. 038	0	1	*	*	Note	*	*	*
--------------	---	---	---	---	------	---	---	---

NOTE2 1 : A full keyboard is used.
0 : A standard keyboard is used.




NOTE3 I/O = selects a device used for data input/output through a reader/punch interface.

Function	Related parameter No.			
	I/O=0	I/O=1	I/O=2	I/O=3
Feed (NFED)	2.7	12.7	50.7	51.7
20 mA current loop (ASR33)	2.2	12.2	Unusable	
Stop bit (STP2)	2.0	12.0	50.0	51.0
I/O device model specification	38.7 38.6	38.7 38.6	38.5 38.4	38.2 38.1
Baud rate	552	553	250	251
Connector number	M5 Channel 1	M5 Channel 1	M74 Channel 2	M77/M73 Channel 3

For M77, either RS-232-C or RS-422 can be selected by parameter No. 55.3.

The connector number is M73 when RS-422 is used with an external clock.

Displaying the directory of floppy disk files

- 1 Press the EDIT switch on the machine operator's panel.
- 2 Press function  key .
- 3 Press soft key **[FLOPPY]**.
- 4 Press page key  or  .
- 5 The screen below appears.

2



DIRECTORY (FLOPPY)		O0555 N0000
NO. FILE NAME		(METER) VOL
0001	PARAMETER	65.6
0002	ALL. PROGRAM	1.9
0003	O0001	1.3
0004	O0002	1.3
0005	O0003	1.3
0006	O0004	1.3
0007	O005	1.3
0008	O0100	1.9
0009	O0555	1.3
19:36:51		S 0 T
EDIT		
[SRHFIL] [READ] [PUNCH] [DELETE] []		

- 6 Press a page key again to display another page of the directory.

Reading files

- 1 Press soft key **[READ]** after directory is displayed.

DIRECTORY (FLOPPY)		O0555 N0000
NO. FILE NAME		(METER) VOL
READ		
_FILE NO. =		PROGRAM NO. =
NUM.		S 0 T
19:38:35		EDIT
[EXEC] [CAN] [] [] [STOP]		

- 2 Enter a file number, then press function  key .
- 3 To modify the program number, enter the program number, then press function  key.
- 4 Press soft key **[EXEC]**.
- 5 Press soft key **[CAN]** to return to the soft key display shown in the screen of directory display.

2. CRT/MDI OPERATION

Outputting programs

- 1 Press soft key **[PUNCH]** after directry is displayed.

DIRECTORY (FLOPPY)		O0555 N0000	
NO. FILE NAME		(METER) VOL	
PUNCH			
_FILE NO. =		_ PROGRAM NO. =	
NUM.		S 0 T	
19:39:17		EDIT	
[EXEC] [CAN] [PRG-NO] [] [STOP]			

- 2 Enter a program number. To write all programs into a single file, enter -9999 in the program number field. In this case, the file name "ALL.PROGRAM" is registered.

Then press function key.

- 3 Press soft key **[EXEC]**.
- 4 Press soft key **[CAN]** to return to the soft key display shown in the screen of directry display.

Deleting files

- 1 Press soft key **[DELETE]** after directry is displayed.

DIRECTORY (FLOPPY)		O0555 N0000	
NO. FILE NAME		(METER) VOL	
DELETE			
_FILE NO. =			
NUM.		S 0 T	
19:39:56		EDIT	
[EXEC] [CAN] [] [] [STOP]			

- 2 Specify the file to be deleted.
When specifying the file with a file number, type the number and press function key.
- 3 Press soft key **[EXEC]**.
- 4 Press soft key **[CAN]** to return to the soft key display shown in the screen of directry display.

Procedure for changing the file name

- 1 Press soft key **[RENAME]** after directry is displayed.
- 2 Position the cursor to FILE NO. then enter the number of the file whose name is to be changed. Press the key.
- 3 Position the cursor to NAME and key in a new file name. Then, press the key.
- 4 Press soft key **[EXEC]**.
- 5 To return to the previous screen, press the **[CAN]** soft key.

2

```
FILE DIRECTORY          O0001 N0000
NO. FILE NAME          (METER) VOL
0001 PARAMETER          87.1
0002 ALL.PROGRAM        87.1
0003 O0001              1.9
0004 O0021              7.1
0005 O0041              7.1
0006 O0615              5.8
0007 O0651              9.1
0008 O0601              7.1
0009 O0645              5.8
RENAME
  FILE NO. =          NAME=
  NUM.          S      0 T0101
  21:59:53      EDIT
[ EXEC ] [ CAN ] [    ] [    ] [ STOP ]
```

3. OPERATION LIST

Classification	Function	KEY SW	SETTING PWE=1	Mode
Reset	Resetting of operating time			—
	Resetting of number of machined parts			—
	Resetting of OT alarm			At power-up
	Resetting of alarm 100			—
Registration from MDI	Parameter input		○	MDI mode
	Offset input			—
	Setting data input			MDI mode
	PMC parameter input	○	○	MDI mode
	Tool length measurement			JOG mode
Registration from NC tape	Parameter input (NC tape → memory)		○	EDIT mode
			○	—
	PMC parameter input	○	○	EDIT mode
	Offset input			EDIT mode
	Program registration	○		EDIT/AUTO mode
Tape punch out	Parameter punch out			EDIT mode
	PMC parameter punch out			EDIT mode
	Offset punch out			EDIT mode
	All programs punch out			EDIT mode
	One program punch out			EDIT mode

Function key	Operation
POS	<input type="button" value="R"/> → <input type="button" value="CAN"/>
POS	<input type="button" value="P"/> → <input type="button" value="CAN"/>
—	<input type="button" value="P"/> AND <input type="button" value="CAN"/>
—	<input type="button" value="CAN"/> AND <input type="button" value="RESET"/>
PARAM	<input type="button" value="No."/> → Parameter number → <input type="button" value="INPUT"/> → Data → <input type="button" value="INPUT"/> →PWE =0 → <input type="button" value="RESET"/>
OFSET	<input type="button" value="No."/> → Offset number → <input type="button" value="INPUT"/> → Offset value→ <input type="button" value="INPUT"/>
PARAM	<input type="button" value="No."/> → <input type="button" value="0"/> → <input type="button" value="INPUT"/> → Data → <input type="button" value="INPUT"/>
DGNOS	<input type="button" value="No."/> → Diagnostic number → <input type="button" value="INPUT"/> → Data → <input type="button" value="INPUT"/>
POS → OFSET	<input type="button" value="POS"/> (Relative coordinate system display) <input type="button" value="Z"/> → <input type="button" value="CAN"/> → <input type="button" value="OFSET"/> → Place tool in measurement position → <input type="button" value="No."/> → Offset number → <input type="button" value="INPUT"/> → <input type="button" value="EOB"/> AND <input type="button" value="Z"/>
PARAM	<input type="button" value="INPUT"/>
PARAM	Emergency stop → <input type="button" value="EOB"/> AND <input type="button" value="INPUT"/>
DGNOS	<input type="button" value="INPUT"/>
OFSET	<input type="button" value="INPUT"/>
PRGRM	<input type="button" value="INPUT"/>
PARAM	<input type="button" value="OUTPT"/>
DGNOS	<input type="button" value="OUTPT"/>
OFSET	<input type="button" value="OUTPT"/>
PRGRM	<input type="button" value="0"/> → -9999 → <input type="button" value="OUTPT"/>
PRGRM	<input type="button" value="0"/> → Program number → <input type="button" value="OUTPT"/>

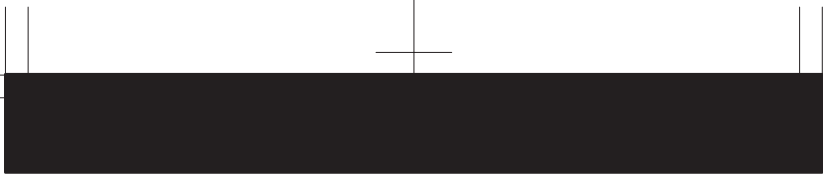
3. OPERATION LIST

Classification	Function	KEY SW	SETTING PWE=1	Mode
Search	Search for program number			EDIT/AUTO mode
	Search for sequence number			AUTO mode
	Search for address/word			EDIT mode
	Search for address only			EDIT mode
	Search for offset number			—
	Search for diagnostic number			—
	Search for parameter number			—
Editing	Display of memory used			EDIT mode
	Deletion of all programs	○		EDIT mode
	Deletion of one program	○		EDIT mode
	Deletion of multiple blocks	○		EDIT mode
	Deletion of one block	○		EDIT mode
	Word deletion	○		EDIT mode
	Word alteration	○		EDIT mode
	Word insertion	○		EDIT mode
Collation	Memory collation			EDIT/AUTO mode
I/O to and from FANUC Cassette	Program registration	○		EDIT/AUTO mode
	Output of all programs			EDIT mode
	Output of one program			EDIT mode
	File head search			EDIT/AUTO mode
	File deletion	○		EDIT mode
	Program collation			EDIT/AUTO mode

Function key	Operation
PRGRM	<input type="text" value="O"/> → Program number → <input type="text" value="⏏"/> (Cursor)
PRGRM	Program number search → <input type="text" value="N"/> → Sequence number search → <input type="text" value="⏏"/> (Cursor)
PRGRM	Data to be searched for → <input type="text" value="⏏"/> (Cursor)
PRGRM	Address to be searched for → <input type="text" value="⏏"/> (Cursor)
OFSET	<input type="text" value="No."/> → Offset number → <input type="text" value="INPUT"/>
DGNOS	<input type="text" value="No."/> → Diagnostic number → <input type="text" value="INPUT"/>
PARAM	<input type="text" value="No."/> → Parameter number → <input type="text" value="INPUT"/>
PRGRM	<input type="text" value="PRGRM"/>
PRGRM	<input type="text" value="O"/> → -9999 → <input type="text" value="DELET"/>
PRGRM	<input type="text" value="O"/> → Program number → <input type="text" value="DELET"/>
PRGRM	<input type="text" value="O"/> → Sequence number → <input type="text" value="DELET"/>
PRGRM	<input type="text" value="EOB"/> → <input type="text" value="DELET"/>
PRGRM	Search for word to be deleted → <input type="text" value="DELET"/>
PRGRM	Search for word to be changed → New data → <input type="text" value="ALTER"/>
PRGRM	Search for word immediately before insertion location → New data → <input type="text" value="INSRT"/>
PRGRM	<input type="text" value="INPUT"/>
PRGRM	<input type="text" value="N"/> → File number → <input type="text" value="INPUT"/> → <input type="text" value="INPUT"/>
PRGRM	<input type="text" value="O"/> → -9999 → <input type="text" value="OUTPT"/>
PRGRM	<input type="text" value="O"/> → Program number → <input type="text" value="OUTPT"/>
PRGRM	<input type="text" value="N"/> → File number → <input type="text" value="INPUT"/>
PRGRM	<input type="text" value="N"/> → File number → <input type="text" value="OUTPT"/>
PRGRM	<input type="text" value="N"/> → File number → <input type="text" value="INPUT"/> → <input type="text" value="INPUT"/>

3. OPERATION LIST

Classifi- cation	Function	KEY SW	S E T- TING PWE=1	Mode
Play-back	NC data input			TEACH-IN JOG/HANDLE mode
Clear	Memory all clear			At power-up
	Parameter clear		○	At power-up
	Program clear		○	At power-up
	Sub PCB all clear			At power-up



Function key	Operation
PRGRM	Move machine → <input type="button" value="X"/> , <input type="button" value="Y"/> or <input type="button" value="Z"/> → <input type="button" value="INSRT"/> → NC data → <input type="button" value="INSRT"/> → <input type="button" value="EOB"/> → <input type="button" value="INSRT"/>
—	<input type="button" value="RESET"/> AND <input type="button" value="DELET"/>
—	<input type="button" value="RESET"/>
—	<input type="button" value="DELET"/>
—	<input type="button" value="DELET"/> AND <input type="button" value="S"/> (However, set PWE on main side to 0.)

3

4. G CODE LIST

M series

G code list (M series)

(1/3)

G code	Group	Function	
G00	01	Positioning	
G01		Linear interpolation	
G02		Circular interpolation/Helical interpolation CW	
G03		Circular interpolation/Helical interpolation CCW	
G04	00	Dwell, Exact stop	
G05		High speed cycle machining	
G08		Advanced preview control	
G09		Exact stop	
G10		Data setting	
G11		Data setting mode cancel	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20	06	Input in inch	
G21		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G27	00	Reference position return check	
G28		Return to reference position	
G29		Return from reference position	
G30		2nd, 3rd and 4th reference position return	
G31		Skip function	
G33	01	Thread cutting	
G37	00	Automatic tool length measurement	
G39		Corner offset circular interpolation	
G40	07	Cutter compensation cancel	
G41		Cutter compensation left	
G42		Cutter compensation right	
G43	08	Tool length compensation + direction	
G44		Tool length compensation – direction	
G45	00	Tool offset increase	
G46		Tool offset decrease	
G47		Tool offset double increase	
G48		Tool offset double decrease	
G49	08	Tool length compensation cancel	
G50	11	Scaling cancel	
G51		Scaling	
G52	00	Local coordinate system setting	
G53		Machine coordinate system selection	

G code list (M series)

(2/3)

G code	Group	Function
G54	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call
G66	12	Macro modal call
G67		Macro modal call cancel
G68	16	Coordinate rotation
G69		Coordinate rotation cancel
G73	09	Peck drilling cycle
G74	09	Counter tapping cycle
G75	01	Plunge grinding cycle (0-GSC)
G76	09	Fine boring cycle
G77	01	Direct constant-dimension plunge grinding cycle (0-GSC)
G78		Continuous-feed surface grinding cycle (0-GSC)
G79		Intermittent-feed surface grinding cycle (0-GSC)
G80	09	Canned cycle cancel/external operation function cancel
G81		Drilling cycle, spot boring cycle or external operation function
G82		Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	03	Absolute command
G91		Increment command
G92	00	Setting for work coordinate system or clamp at maximum spindle speed
G94	05	Feed per minute
G95		Feed per rotation
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G98	10	Return to initial point in canned cycle
G99		Return to R point in canned cycle
G107	00	Cylindrical interpolation

4. G CODE LIST

G code list (M series)

(3/3)

G code	Group	Function
■ G150	19	Normal direction control cancel mode
G151		Normal direction control left side on
G152		Normal direction control right side on
■ G160	20	In-feed control function cancel (0-GSC)
G161	20	In-feed control function (0-GSC)

CAUTION

- Multiple G codes of different groups can be specified in a single block. When multiple G codes of one group are specified in a block, the G code specified last is effective.
- If any G code of group 01 is specified in a canned cycle mode, the canned cycle is automatically cancelled and the G80 condition is entered. However, a G code of group 01 is not affected by any of the canned cycle G codes.

NOTE

- Modal G codes have the following initial conditions when the power is turned on or the system is reset to the clear state (bit 6 of parameter No. 045).
 - Those G codes marked ■ in Table 3 are specified automatically.
 - G20 and G21 retain their original conditions.
 - When the power is turned on, G22 is specified automatically. When the system is reset, G22 and G23 retain their original conditions.
 - G00 or G01 is automatically selected depending on the setting of bit 6 of parameter No. 011.
 - G90 or G91 is automatically selected depending on the setting of bit 7 of parameter No. 030.
- The G codes of group 00, except G10 and G11, are one-shot G codes.
- If a G code that does not appear in the G code list is specified, or a G code whose options are not supported is specified, alarm No. 010 is displayed.
- A G code is displayed from each group.

T series

G code list (T series)

(1/2)

G code system (Note 7)			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (rapid traverse)
G01	G01	G01		Linear interpolation (feed)
G02	G02	G02		Circular interpolation CW
G03	G03	G03		Circular interpolation CCW
G04	G04	G04		Dwell
G10	G10	G10	00	Data setting
G11	G11	G11		Data setting mode cancel
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Inch data input
G21	G21	G71		Metric data input
G22	G22	G22	09	Stored stroke check function ON
G23	G23	G23		Stored stroke check function OFF
G25	G25	G25	08	Spindle speed fluctuation detect OFF
G26	G26	G26		Spindle speed fluctuation detect ON
G27	G27	G27	00	Reference point return check
G28	G28	G28		Return to reference point
G30	G30	G30		2nd, 3rd, 4th reference point return
G31	G31	G31		Skip cutting
G32	G33	G33	01	Thread cutting
G34	G34	G34		Variable-lead thread cutting
G36	G36	G36	00	Automatic tool compensation X
G37	G37	G37		Automatic tool compensation Z
G40	G40	G40	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation left
G42	G42	G42		Tool nose radius compensation right
G50	G92	G92	00	Coordinate system setting, max. spindle speed setting
G52	G52	G52		Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordinate system 1 setting
G55	G55	G55		Workpiece coordinate system 2 setting
G56	G56	G56		Workpiece coordinate system 3 setting
G57	G57	G57		Workpiece coordinate system 4 setting
G58	G58	G58		Workpiece coordinate system 5 setting
G59	G59	G59		Workpiece coordinate system 6 setting
G65	G65	G65	00	Macro calling
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G68	G68	G68	04	Mirror image for double turrets ON or balance cut mode (0-TTC)
G69	G69	G69		Mirror image for double turrets OFF or balance cut mode cancel (0-TTC)

4. G CODE LIST

G code list (T series)


(2/2)

G code system (Note 7)			Group	Function
A	B	C		
G70	G70	G72	00	Finishing cycle (other than 0–GCC or 00–GCC)
G71	G71	G73		Stock removal in turning (other than 0–GCC or 00–GCC)
G72	G72	G74		Stock removal in facing (other than 0–GCC or 00–GCC)
G73	G73	G75		Pattern repeating (other than 0–GCC or 00–GCC)
G74	G74	G76		Peck drilling on Z axis (other than 0–GCC or 00–GCC)
G75	G75	G77		Grooving on X axis (other than 0–GCC or 00–GCC)
G76	G76	G78		Multiple threading cycle (other than 0–GCC or 00–GCC)
G71	G71	G72	01	Traverse grinding cycle (0–GCC, 00–GCC)
G72	G72	G73		Traverse direct constant dimension grinding cycle (0–GCC, 00–GCC)
G73	G73	G74		Oscillation grinding cycle (0–GCC, 00–GCC)
G74	G74	G75		Oscillation direct constant–dimension grind- ing cycle (0–GCC, 00–GCC)
G80	G80	G80	10	Canned cycle for drilling cancel
G83	G83	G83		Cycle for face drilling
G84	G84	G84		Cycle for face tapping
G86	G86	G86		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		Endface turning cycle
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G98	G94	G94	05	Per minute feed
G99	G95	G95		Per revolution feed
—	G90	G90	03	Absolute programming
—	G91	G91		Incremental programming
—	G98	G98	11	Return to initial level
—	G99	G99		Return to R point level
G107	G107	G107	00	Cylindrical interpolation
G112	G112	G112	21	Polar coordinate interpolation mode
G113	G113	G113		Polar coordinate interpolation cancel mode
G250	G250	G250	20	Polygonal turning cancel
G251	G251	G251		Polygonal turning

CAUTION



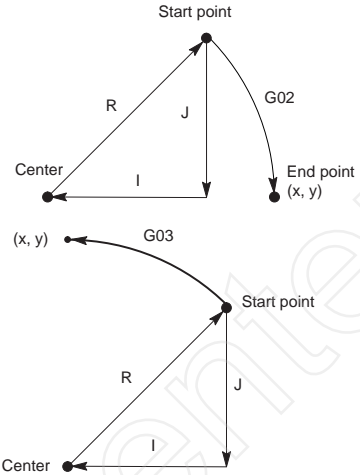
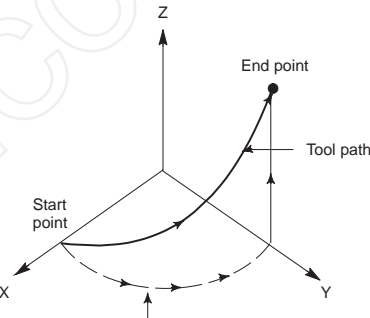
1. A number of G codes can be specified in the same block. When more than one G code of the same group is specified, the G code specified later is effective.
2. If any G code of group 01 is specified in a canned cycle mode, the canned cycle is automatically cancelled and the G80 condition is entered. However a G code of group 01 is not affected by any of the canned cycle G codes.

NOTE

1. G codes marked  are initial G codes when turning power on. For G20 and G21, the G code before turning power off remains. G00 or G01 can be selected by parameter setting.
2. G codes of group 00 are not modal. They are only effective in the block in which they are specified.
3. If a G code not listed on the table of G codes is inputted, or optional G code not specified in the system is commanded, an alarm (No. 010) is displayed.
4. A G code is displayed from each group.
5. G code system B and C are options. Whether G code system B or C is set by parameter No. 0036: GSPC.

4

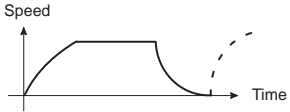
5. PROGRAM FORMAT

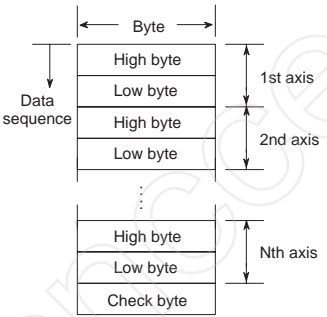
Functions	Explanation
Positioning (G00)	
Linear interpolation (G01)	
Circular interpolation (G02, G03)	
Helical cutting (G02, G03)	 <p>The feedrate along the circumference of two circular interpolated axes is the specified feedrate.</p>

Tape format	T series	M series
G00P__;	○	○
G01P__;	○	○
G17 $\left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\}$ Xp__Yp__ $\left\{ \begin{array}{c} R_ \\ I_J_ \end{array} \right\}$ F__; G18 $\left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\}$ Xp__Zp__ $\left\{ \begin{array}{c} R_ \\ I_K_ \end{array} \right\}$ F__; G19 $\left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\}$ Yp__Zp__ $\left\{ \begin{array}{c} R_ \\ I_K_ \end{array} \right\}$ F__;	○	○
Synchronously with arc of XpYp plane G17 $\left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\}$ Xp__Yp__ $\left\{ \begin{array}{c} I_J_ \\ R_ \end{array} \right\}$ α__ F__; Synchronously with arc of ZpYp plane G18 $\left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\}$ Xp__Zp__ $\left\{ \begin{array}{c} I_K_ \\ R_ \end{array} \right\}$ α__ F__; Synchronously with arc of YpZp plane G19 $\left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\}$ Yp__Zp__ $\left\{ \begin{array}{c} J_K_ \\ R_ \end{array} \right\}$ α__ F__; Any one axis where circular interpolation is not applied.		○

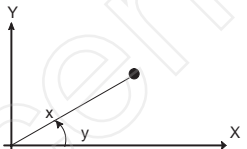

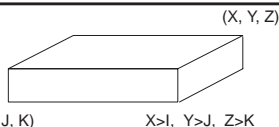
5

5. PROGRAM FORMAT

Functions	Explanation
Dwell (G04)	(Example) G04 P1000; Dwell by 1 seconds
Exact stop (G04, G09)	
High-speed cycle machining (G05)	(Example) Cycle 1: connection 2, repetition 1 Cycle 2: connection 3, repetition 3 Cycle 3: connection 0, repetition 1 G05P10001L2; Cycle is executed as 1, 2, 2, 2, 3, 1, 2, 2, 2, 3
High-speed remote buffer A (G05)	Specify G05 only in a block using normal NC command format. Then specify move data in the special format explained next page. When zero is specified as the travel distance along all axes, normal NC command format can be used again for subsequent command specification.
High-speed remote buffer B (G05)	High-speed remote buffer A uses binary data. On the other hand, high-speed remote buffer B can directly use NC language coded with equipment such as an automatic programming unit to perform high-speed machining.
Advanced preview control (G08)	This function can minimize the delay caused by acceleration/deceleration, which increases together with the feedrate, as well as the delay in the servo system. Tool movement can thus faithfully follow the command values, reducing the degree of error in the machined figure.

Tape format	T series	M series
G04 { X_ } ; Dwell by second P_ }	<input type="radio"/>	<input type="radio"/>
G04 ; G09P ;		<input type="radio"/>
G05 P10 xxxLAAA ; xxx : Start program number of called programs AAA : Repetition time of machining cycle	<input type="radio"/>	<input type="radio"/>
Binary input operation enabled: G05; Binary input operation disabled: The travel distance along all axes are set to zero. Data format for binary input operation  All data must be specified in binary.	<input type="radio"/>	<input type="radio"/>
G05P01; Start high-speed machining X_Y_Z_ ; G05P00; End high-speed machining		<input type="radio"/>
G08 Pp; p=1: Advanced preview control mode on p=0: Advanced preview control mode off		<input type="radio"/>

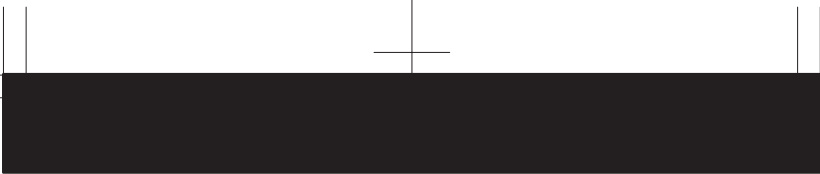
5. PROGRAM FORMAT

Functions	Explanation
Change of offset value by program (G10)	<p>The tool compensation amount can be set or changed with the G10 command.</p> <p>When G10 is used in absolute input (G90), the compensation amount specified in the command becomes the new tool compensation amount.</p> <p>When G10 is used in incremental input (G91), the compensation amount specified in the command is added to the amount currently set.</p>
Change of offset value by program (G10)	
Change of parameter by program (G10)	The parameter value can be changed by the machining program.
Polar coordinate command mode (G15, G16)	
XpYp plane selection (G17) ZpXp plane selection (G18) YpZp plane selection (G19)	
Inch/metric conversion (G20, G21)	
Extended stored stroke limit check on (G22, G23)	

Tape format	T series	M series
G10P__X (U)__Y (V)__Z (W)__R (C)__Q__; For geometry offset amount P=10000+geometry offset number For offset amount P= wear offset number R : Tool nose radius offset value Q : Imaginary tool nose number	<input type="radio"/>	
G10L__PpRr; p : Offset No. r : Tool compensation amount Format (1) For tool compensation memory A G10 L11 P__R__; (2) For tool compensation memory B Setting/changing the geometric compensation amount G10 L10 P__R__; Setting/changing the wear compensation amount G10 L11 P__R__;		<input type="radio"/>
G10 L50 ; N__P__ ; G11 N : Parameter number P : Parameter value	<input type="radio"/>	<input type="radio"/>
G16 ; Polar coordinate command X _x Y _y ; x: radius, y: angle (*) G15 ; Polar coordinate command cancel		<input type="radio"/>
G17 ; G18 ; G19 ;	<input type="radio"/>	<input type="radio"/>
G20 ; Inch input G21 ; Metric input	<input type="radio"/>	<input type="radio"/>
G22X__Y__Z__I__J__K__ ; on G23 ; off (X, Z, I and K only for T series)	<input type="radio"/>	<input type="radio"/>

5. PROGRAM FORMAT

Functions	Explanation
Spindle speed fluctuation detection on (G26) Spindle speed fluctuation detection off (G25)	
(Example)	
(1) When an alarm is raised after a specified spindle speed is reached	
Spindle speed	
(2) When an alarm is raised before a specified spindle speed is reached	
Spindle speed	
Reference position return check (G27)	



Tape format	T series	M series
<p>G26PpQqRr ; spindle fluctuation detection on</p> <p>p: Time (in ms) from the issue of a new spindle rotation command (S command) to the start of checking whether the actual spindle speed is so fast that an overheat can occur. (When a specified speed is reached within the time period of P, a check is started at that time.)</p> <p>q: Tolerance (%) of a specified spindle speed (If a specified spindle speed lies within this range, it is regarded as having reached the specified value. Then, the checking of an actual spindle speed is started.)</p> <p>$q = (1 - \text{actual spindle speed} / \text{specified spindle speed}) \times 100$</p> <p>r: Spindle speed fluctuation (%) at which the actual spindle speed is so fast that an overheat can occur</p> <p>$r = (1 - \text{speed that can cause overheat} / \text{specified spindle speed}) \times 100$</p> <p>G26 enables the spindle speed fluctuation detection function, and G25 disables the spindle speed fluctuation detection.</p> <p>G25 ; Spindle fluctuation detection off</p>	<p>○</p>	
G27P__ ;	<p>○</p>	<p>○</p>

5

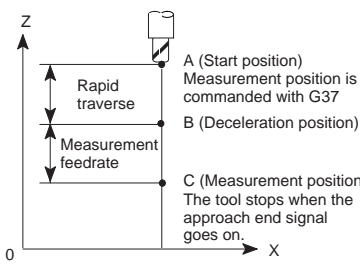
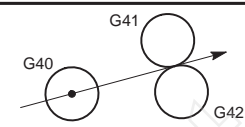
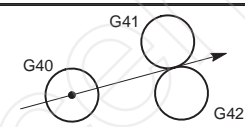
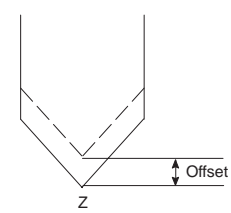


5. PROGRAM FORMAT

Functions	Explanation
Reference position return (G28) 2nd, 3rd, 4th reference position return (G30)	<p>Reference position</p> <p>Intermediate point</p> <p>Start point</p>
Return to reference position return start position (G29)	<p>Reference position</p> <p>Intermediate position</p> <p>P</p>
Skip function (G31)	<p>Skip signal</p> <p>Start point</p> <p>P</p>
Multi-step skip function (G31)	
Equal lead thread cutting (G32)	<p>F</p> <p>F: Lead</p>
Thread cutting (G33)	
Variable lead thread cutting	
Automatic tool compensation (G36, G37)	<p>Starting point</p> <p>Specified position (Xa or Za)</p> <p>Measured position reach signal</p> <p>Measured position</p> <p>Offset value set by this command</p>

Tape format	T series	M series
G28P__ ; G30 { P2 } P__ ; P2: 2nd reference position return { P3 } P3: 3rd reference position return { P4 } P4: 4th reference position return	<input type="radio"/>	<input type="radio"/>
G29P__ ;		<input type="radio"/>
G31P__F__ ;	<input type="radio"/>	<input type="radio"/>
Move command G31 P__F__P__ ; F__ : Feedrate P__ : P1-P4 Dwell G04X (U, P)__(Q__); X(U, P)__: Dwell time Q__ : Q1-Q4	GCC	
G32P__F__ ;	<input type="radio"/>	
G33P__F__ ;		<input type="radio"/>
G34P__F{K} ; f: Longer axis lead at the start position k: increase/decrease value per spindle revolution	<input type="radio"/>	
G36X_xa ; G37Z_za ; X xa, Z za: Specified position	<input type="radio"/>	

5. PROGRAM FORMAT

Functions	Explanation
Automatic tool length measurement (G37)	 <p>A (Start position) Measurement position is commanded with G37 B (Deceleration position) C (Measurement position) The tool stops when the approach end signal goes on.</p> <p>0 X</p> <p>Compensation value = (Current compensation value) + [(Coordinates of the point at which the tool is stopped) - (Coordinates of the programmed measurement position)]</p>
Tool nose radius compensation (G40, G41, G42)	 <p>G40 : Programmed path G41 : Left of programmed path G42 : Right of programmed path</p>
Cutter compensation B (G39 to G42) Cutter compensation (G40 to G42)	 <p>G40 : Programmed path G41 : Left of programmed path G42 : Right of programmed path</p>
Tool length compensation A, B, C (G43, G44, G49)	 <p>G43: + offset G44: - offset</p> <p>Z</p> <p>Offset</p>


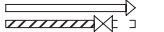
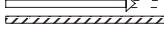
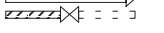

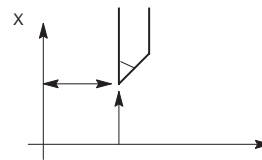
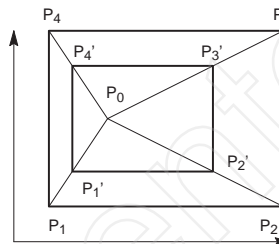
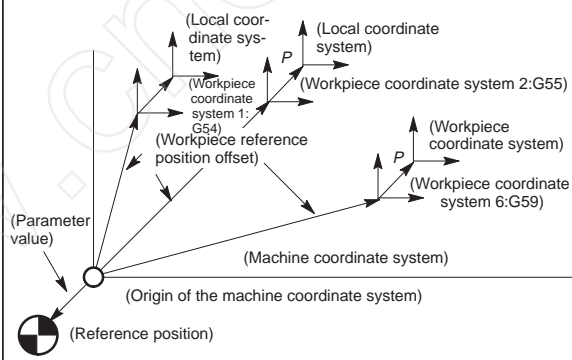


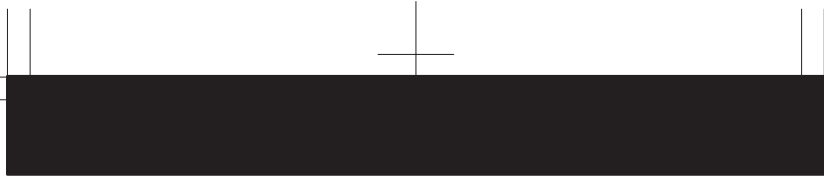
Tape format	T series	M series
<p>G92P__ ; Sets the workpiece coordinate system. (It can be set with G54 to G59.)</p> <p>H__ ; Specifies an offset number for tool length offset.</p> <p>G90 G37P__ ; Absolute command .G37 is valid only in the block in which it is specified. P__ indicates the X-, Y-, Z-, or fourth axis.</p>		○
$\begin{Bmatrix} G40 \\ G41 \\ G42 \end{Bmatrix} P_ ;$	○	
<p>G39X(I)Y(J) ; Corner offset circular interpolation (Cutter compensation B only)</p> $\begin{Bmatrix} G17 \\ G18 \\ G19 \end{Bmatrix} \begin{Bmatrix} G40 \\ G41 \\ G42 \end{Bmatrix} D(H)_ ;$ <p>D(H): Tool offset number</p>		○
$\begin{Bmatrix} G43 \\ G44 \end{Bmatrix} (Z_)H_ ; \text{ Tool length compensation A}$ $\begin{Bmatrix} G17 \\ G18 \\ G19 \end{Bmatrix} \begin{Bmatrix} G43 \\ G44 \end{Bmatrix} \left(\begin{Bmatrix} X_ \\ Y_ \\ Z_ \end{Bmatrix} \right) H_ ;$ <p>Tool length compensation B $\begin{Bmatrix} G43 \\ G44 \end{Bmatrix} \alpha_ H_ ;$ Tool length compensation C</p> <p>H: Offset number α: Arbitrary one axis G49 ; Tool length compensation cancel</p>		○

5



5. PROGRAM FORMAT

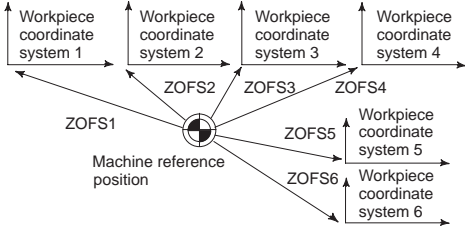
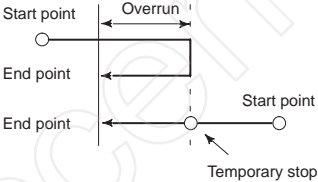
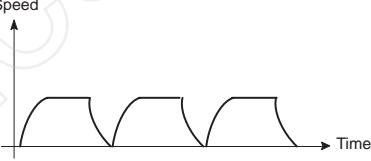
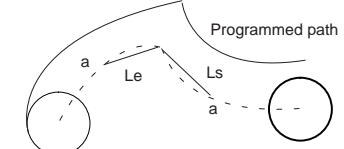
Functions	Explanation
Tool offset (G45 – G48)	<div> G45  Increases G46  Decrease G47  Double increase G48  Double decrease  : offset value P </div>
Coordinate system setting Spindle speed setting (G50)	
Scaling (G50, G51)	 <p> P_1 to P_4: Programmed shape P_1' to P_4': Scaled shape P_0: Scaling center </p>
Local coordinate system setting (G52)	
Machine coordinate system selection (G53)	

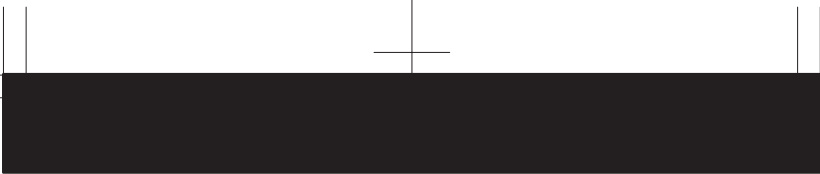


Tape format	T series	M series
$\left\{ \begin{array}{l} \text{G45 (increase)} \\ \text{G46 (decrease)} \\ \text{G47 (double increase)} \\ \text{G48 (double decrease)} \end{array} \right\} \left\{ \begin{array}{l} \text{X} \\ \text{Y} \\ \text{Z} \end{array} \right\} \text{Dxx ;}$		<input type="radio"/>
G50X__Z__ ; Coordinate system setting G50S__ ; Spindle speed setting	<input type="radio"/>	
G51X__Y__Z__P__(or I__J__K__) ; G50 ; Cancel X, Y, Z: Scaling center P: Magnification (I, J, and K are the scaling magnifications for the X-, Y-, and Z-axes respectively.)		<input type="radio"/>
G52P__ ; Local coordinate system setting G52P0 ; Local coordinate system cancel	<input type="radio"/>	<input type="radio"/>
G53P__ ;	<input type="radio"/>	<input type="radio"/>

5

5. PROGRAM FORMAT

Functions	Explanation
Work coordinate system 1 – 6 selection (G54 – G59)	 <p>ZOFS1: Reference position offset for workpiece coordinate system 1 ZOFS2: Reference position offset for workpiece coordinate system 2 ZOFS3: Reference position offset for workpiece coordinate system 3 ZOFS4: Reference position offset for workpiece coordinate system 4 ZOFS5: Reference position offset for workpiece coordinate system 5 ZOFS6: Reference position offset for workpiece coordinate system 6</p>
Additional work coordinate system selection (G54P)	(Example) G54P12 ; Selecting additional work coordinate system 12
Spindle direction positioning (G60)	
Exact stop mode (G61)	
Automatic corner override (G62)	 <p>Override is applied from a to b</p>

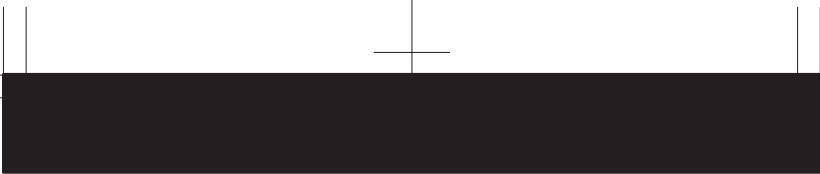


Tape format	T series	M series
G54P__ ; Work coordinate system 1 selection G55P__ ; Work coordinate system 2 selection G56P__ ; Work coordinate system 3 selection G57P__ ; Work coordinate system 4 selection G58P__ ; Work coordinate system 5 selection G59P__ ; Work coordinate system 6 selection	<input type="radio"/>	<input type="radio"/>
G54Pn ; (n=1 – 64)		<input type="radio"/>
G60P__ ;		<input type="radio"/>
G61 ;		<input type="radio"/>
G62 ;		<input type="radio"/>

5

5. PROGRAM FORMAT

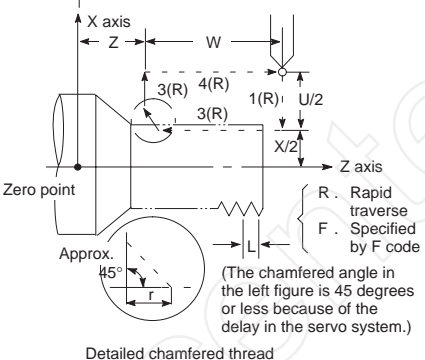
Functions	Explanation
Tapping mode (G63)	<p>Speed</p> <p>Time</p>
Cutting mode (G64)	<p>Speed</p> <p>Time</p>
Macro call (G65)	<p>Main program</p> <pre>O_: G66P0001L_: M02;</pre> <p>Macro program</p> <pre>O0001; M99</pre>
Macro model call (G66, G67)	<p>Main program</p> <pre>O_: G66P0001L_: X_: Y_: G67;</pre> <p>Macro program</p> <pre>O0001; M99</pre>
Mirror image for double turrets (G68, G69)	
Coordinate rotation (G68, G69)	

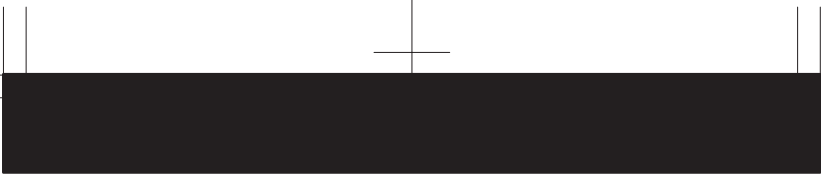


Tape format	T series	M series
G63 ;		<input type="radio"/>
G64 ;		<input type="radio"/>
G65 P__L__ ; P: Program number L: Repetition count (1 to 9999)	<input type="radio"/>	<input type="radio"/>
G66 P__L__ ; G67 ; Cancel P: Program number L: Repetition count (1 to 9999)	<input type="radio"/>	<input type="radio"/>
G68 ; Mirror image for double turrets G69 ; Mirror image cancel	TT	
$\left\{ \begin{matrix} G17 \\ G18 \\ G19 \end{matrix} \right\} G68 \alpha_ \beta_ R_ ;$ G69 ; α, β : 3 axes corresponding to G17, G18, G19 R: Rotation angle		<input type="radio"/>

5

5. PROGRAM FORMAT

Functions	Explanation
Canned cycle for lathes (G70 to G76) (G90, G92, G94)	<p>Canned cycle</p> <p>G90: Outer diameter/internal diameter cutting cycle</p> <p>G92: Thread cutting cycle</p> <p>G94: End face turning cycle</p> <p>Multiple repetitive cycle</p> <p>G70: Finishing</p> <p>G71: Stock removal in turning</p> <p>G72: Stock removal in facing</p> <p>G73: Pattern repeating</p> <p>G74: End face peck drilling cycle</p> <p>G75: Outer diameter/internal diameter drilling cycle</p> <p>G76: Multiple thread cutting cycle</p> <p>(Example) G92</p>  <p>The diagram illustrates the G92 thread cutting cycle. It shows a lathe workpiece with a thread being cut. The X-axis is vertical and the Z-axis is horizontal. The zero point is marked on the Z-axis. The thread is defined by parameters: W (total length), U/2 (thread depth), X/2 (thread width), and R (radius). The thread is shown in a detailed view with a 45-degree chamfered angle and a radius r. The text indicates that the chamfered angle is approximately 45 degrees or less due to servo system delay.</p> <p>R . Rapid traverse F . Specified by F code</p> <p>(The chamfered angle in the left figure is 45 degrees or less because of the delay in the servo system.)</p> <p>Detailed chamfered thread</p>



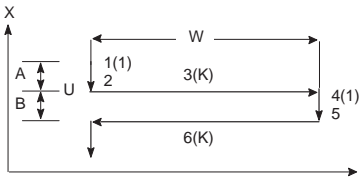
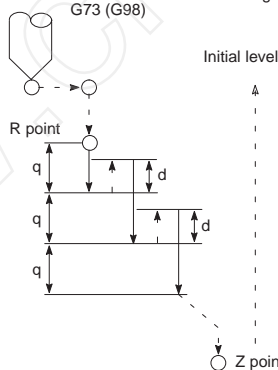
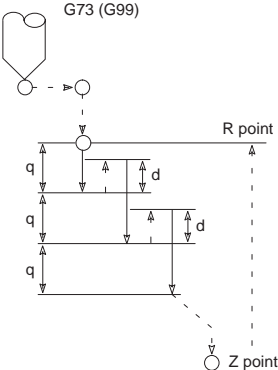
Tape format	T series	M series
G70 P_Q_ ; G71 U_R_ ; G71 P_Q_U_W_F_S_T_ ; G72 W_R_ ; G72 P_Q_U_W_F_S_T_ ; G73 W_R_ ; G73 P_Q_U_W_F_S_T_ ; G74 R_ ; G74 X(u)_Z(w)_P_Q_R_F_ ; G75 R_ ; G75 X(u)_Z(w)_P_Q_R_F_ ; G76 R_ ; G76 X(u)_Z(w)_P_Q_R_F_ ; { G90 } { G92 } X_Z_I_F_ ; G94 X_Z_I_F_ ;	○	

5

www.cnccenter.com

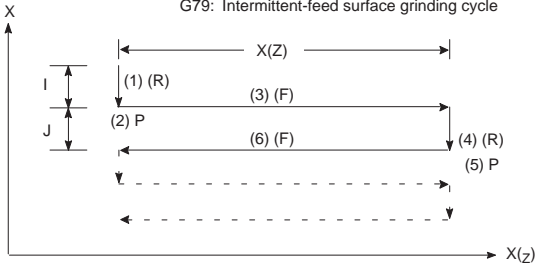
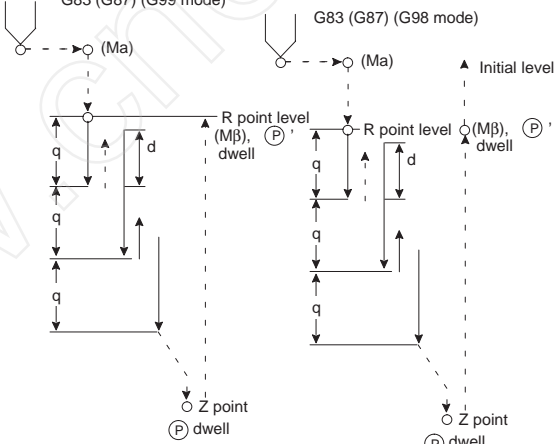


5. PROGRAM FORMAT

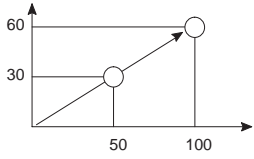
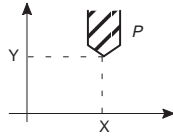
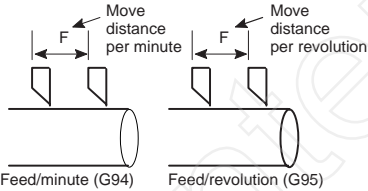
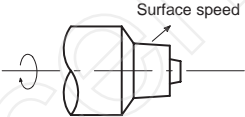
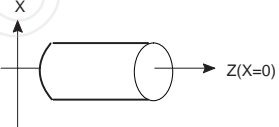
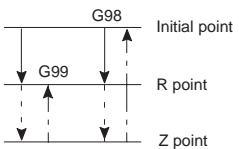
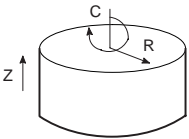
Functions	Explanation
Canned cycle for grinding (G71 – G74)	<p>G71: Traverse grinding cycle G72: Traverse direct fixed-dimension grinding cycle G73: Oscillation grinding cycle G74: Oscillation direct fixed-dimension grinding cycle</p> <p>(Example) G71 G71 A_B_W_U_I_K_H_;</p>  <p>A: First depth of cut B: Second depth of cut W: Grinding range U: Dwell time Maximum specification time: 99999.999 seconds I: Feedrate of A and B K: Feedrate of W H: Number of repetitions Setting value: 1 to 9999</p>
Canned cycle (G73, G74, G80 – G89)	<p>G73: High-speed peck drilling cycle G74: Left-hand tapping cycle G76: Fine boring cycle G80: Cancel G81: Drilling cycle, spot drilling cycle G82: Drilling cycle, counter boring cycle G83: Peck drilling cycle G84: Tapping cycle G85: Boring cycle G86: Boring cycle G87: Boring cycle/Back boring cycle G88: Boring cycle G89: Boring cycle</p> <p>(Example) G73 (G98)</p>  <p>G73 (G99)</p> 

Tape format		T series	M series
G71 A _ B _ W _ U _ I _ K _ H _ ; G72 P _ A _ B _ W _ U _ I _ K _ H _ ; G73 A _ (B _) W _ U _ I _ K _ H _ ; G74 P _ A _ (B _) W _ U _ I _ K _ H _ ;		GCC	
G73 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G74 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G76 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G81 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G82 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G83 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G84 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G85 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G86 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G87 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G88 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G89 X _ Y _ Z _ P _ Q _ R _ F _ K _ ; G80 ; Cancel G <input type="checkbox"/> <input type="checkbox"/> X _ Y _ Z _ R _ Q _ P _ F _ K _ ; <div style="display: flex; justify-content: space-around; margin-top: 10px;"> <div style="text-align: center;"> {X Y Z} Hole position data </div> <div style="text-align: center;"> {P F} Drilling data </div> <div style="text-align: center;"> {K} Number of repeat </div> </div> <div style="display: flex; justify-content: space-around; margin-top: 10px;"> <div style="text-align: center;"> <input type="checkbox"/> Drilling mode </div> </div>		○	
Item	Address	Explanation	
Drilling mode	G <input type="checkbox"/> <input type="checkbox"/>	G73, G74, G76, G80 – G89	
Hole position data	X,Y	Specifies the hole position by an incremental or absolute value.	
Drilling data	Z	Specifies the distance from point R to the bottom of the hole.	
	R	Specifies the distance from the initial level to point R.	
	Q	Specifies each cut-in value with G73 and G83 or the shift value with G76 and G87. (Always specified with an incremental value.)	
	P	Specifies the dwell time at the bottom of the hole.	
	F	Specifies the feed rate.	
Number of repeats	K	Specifies the number of repeats for a series of operation 1 to 6.	

5. PROGRAM FORMAT

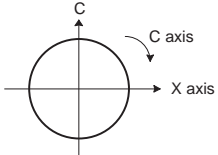
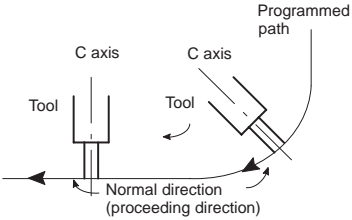
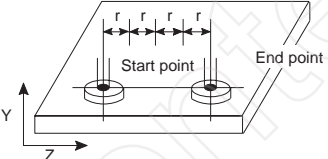
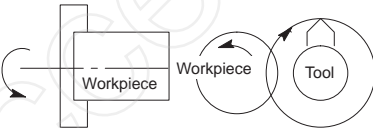
Functions	Explanation
Canned grinding cycle (G75, G77, G78, G79)	G75: Plunge grinding cycle G77: Direct constant-dimension plunge grinding cycle G78: Continuous-feed surface grinding cycle G79: Intermittent-feed surface grinding cycle
(Example)	 <p>G75 I _ J _ K _ X (Z) _ R _ F _ P _ L _ ;</p> <p>I: Depth-of-cut 1 (A sign in the command specifies the direction of cutting.) J: Depth-of-cut 2 (A sign in the command specifies the direction of cutting.) K: Total depth of cut X (Z): Range of grinding (A sign in the command specifies the direction of grinding.) R: Feedrate for I and J F: Feedrate for X (Z) P: Dwell time L: Grinding-wheel wear compensation (Note 1)</p>
Canned cycle for drilling (G80 to G89)	G83: Front drilling cycle G84: Front tapping cycle G85: Front boring cycle G87: Side drilling cycle G88: Side tapping cycle G89: Side boring cycle G80: Drilling cycle cancel
(Example)	 <p>G83 (G87) (G99 mode) G83 (G87) (G98 mode)</p>

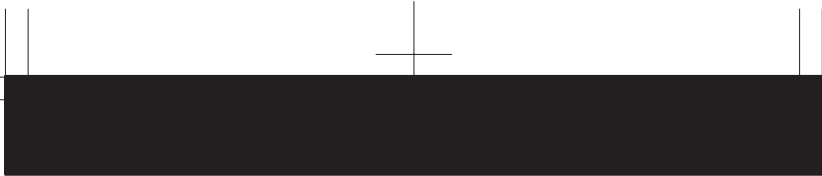
5. PROGRAM FORMAT

Functions	Explanation
Absolute/Incremental command (G90/G91)	<p>(Example)</p>  <p>Absolute command G90X100Y60 ; Incremental command G91X50Y30 ;</p>
Change of workpiece coordinate system (G92)	
Feed/minute, Feed/revolution (G94, G95)	
Constant surface speed control (G96, G97)	
Constant surface speed control (G96, G97)	
Initial point return/R point return (G98, G99)	
Cylindrical interpolation (G107)	

Tape format	T series	M series
G90__ ; Absolute command G91__ ; Incremental command G90__G91__ ; Change of workpiece coordinate system	<input type="radio"/> (G code system B, C)	<input type="radio"/>
G92P__ ;		<input type="radio"/>
G94 F__ ; G98 F__ ; G95 F__ ; or G99 F__ ; (T series (T series <G code system B, C> <G code system A> M series)	<input type="radio"/>	<input type="radio"/>
G96 S _s ; G97 ; Cancel s: Surface speed (m/min or feet/min)	<input type="radio"/>	
G96 S _s P _α ; G97 ; Cancel α: 1 – 3 (X axis, Y axis, Z axis) s: Surface speed (m/min or feet/min)		<input type="radio"/>
G98__ ; G99__ ;	<input type="radio"/> (G code system B, C)	<input type="radio"/>
G107 Cr; Cylindrical interpolation mode C: Rotary axis name r: Radius of cylinder G107 CO; Cylindrical interpolation mode cancel	<input type="radio"/>	<input type="radio"/>

5. PROGRAM FORMAT

Functions	Explanation
Polar coordinate interpolation mode (G112, G113)	 <p>A diagram showing a circle with a horizontal X axis and a vertical C axis. The C axis is labeled with an arrow pointing upwards, and the X axis is labeled with an arrow pointing to the right.</p>
Normal direction control (G140, G141, G142)	 <p>A diagram illustrating normal direction control. It shows a tool moving along a programmed path. The C axis is indicated for both the tool and the path. The normal direction (proceeding direction) is shown as a curved arrow perpendicular to the path.</p>
Infeed control (G160, G161)	<p>(Example)</p>  <p>A diagram showing an example of infeed control. It depicts a workpiece with a start point and an end point. The Y and Z axes are shown. The infeed distance is indicated by multiple small arrows labeled 'r' between the start and end points.</p>
Polygon turning (G250, G251)	 <p>A diagram illustrating polygon turning. It shows a workpiece being turned by a tool. The workpiece is labeled 'Workpiece' and the tool is labeled 'Tool'. The diagram shows the tool moving around the workpiece to create a polygonal shape.</p>



Tape format	T series	M series
G112 ; Polar coordinate interpolation mode G113 ; Polar coordinate interpolation mode cancel	<input type="radio"/>	<input type="radio"/>
G140 ; Normal direction control cancel G141 ; Normal direction control left G142 ; Normal direction control right		<input type="radio"/>
G160 ; Cancel G161 R _L ; r: Cut in depth		0-GSC
G250 ; Polygon turning cancel G251P__Q__ ; Polygon turning P, Q: Potation ratio of spindle and workpiece	<input type="radio"/>	

5

www.cnccenter.com



5. PROGRAM FORMAT

FS0-TC/FS15-TA format comparison table

Function comparison

- A : Both models have the same functions and formats.
 B : Both models have the same functions, but different formats.
 C : Both models have the same formats, but different functions.
 D : The models have different functions and formats.

Item	Function	FS15-TA
1	Positioning	G00X—Z—;
2	Linear interpolation	G01X—Z—F—;
3	Circular interpolation	G02/03X—Z—R—(I—K—)F—;
4	Dwell	G98G04P—/X—/U—; (in time units) G99G04P—/X—/U—; (in rotation units)
5	Cylindrical interpolation	G07.1 Rotation axis name and cylinder diameter
6	Data setting	G10L11P—X—Z—Y—R—Q—; L11 Wear compensation G10L10P—X—Z—Y—R—Q—; L10 Geometric compensation
7	Polar coordinate interpolation mode	G12.1
8	Polar coordinate interpolation cancel mode	G13.1
9	Plane selection	G17/G18/G19
10	Inch input	G20
11	Metric input	G21
12	Stored stroke check ON	G22X—Z—I—K—;
13	Stored stroke check OFF	G23;
14	Spindle speed fluctuation detection OFF	G25;
15	Spindle speed fluctuation detection ON	G26P—Q—R—D—; P : Check start timer Q : Check start tolerance ratio R : Minimum variation ratio detected as an alarm D : Minimum variation width detected as an alarm
16	Reference position return check	G27X—Z—;
17	1st reference position return check	G28X—Z—;
18	nth reference position return check	G30P2/3/4X—Z—;
19	Skip cutting	G31X—Z—;
20	Thread cutting	G32X—Z—F—Q—; Q : Threading start shift angle G32X—Z—E—Q—; E : Number of screw threads per inch

FS0-TC	FS0-TC tape format conversion	Function comparison
G00X—Z—;		C
G01X—Z—F—;		A
G02/03X—Z—R—(I—K—)F—;		C
G04P—/X—/U—; (in time units)		C
G107 Rotation axis name and cylinder di- ameter		B
G10P—X—Z—Y—R—Q—; For wear compensation, P : Wear compensation num- ber For geometric compensation, P : 10000 + geometric com- pensation number		B
G112		B
G113		B
G17/G18/G19		A
G20		C
G21		C
G22X—Z—I—K—;		A
G23;		A
G25;		A
G26P—Q—R—; P : Check start timer Q : Check start tolerance ratio R : Minimum variation ratio de- tected as an alarm		C
G27X—Z—;		A
G28X—Z—;		A
G30P2/3/4X—Z—;		A
G31X—Z—;		A
G32X—Z—F—;	G32X—Z—E— E : Lead along the ma- jor axis. It is impossible to specify the number of screw threads per inch and the threading start shift angle.	D

5. PROGRAM FORMAT

Item	Function	FS15-TA
21	Variable lead thread cutting	G34X—Z—F—K—;
22	Automatic tool offset	G37.*X/Z—;
23	Tool nose radius compensation cancel	G40
24	Tool nose radius compensation (Left)	G41
25	Tool nose radius compensation (Right)	G42
26	Coordinate system setting	G50X—Z—;
27	Polygon turning cancel	G50.2
28	Polygon turning	G51.2P—Q—;
29	Local coordinate system setting	G52X—Z—;
30	Machine coordinate system select	G53X—Z—;
31	Workpiece coordinate system select	G54 to G59
32	Macro calling	G65P—L—Argument;
33	Macro modal calling	G66P—L—Argument;
34	Macro modal calling cancel	G67;
35	Mirror image ON for double turret	G68;
36	Mirror image OFF for double turret	G69;
37	Finishing cycle	G70P—Q—;
38	Outer surface rough cutting canned cycle	G71P—Q—U—W—I—K—D—F—S—T—;
39	End face rough cutting canned cycle	G72P—Q—U—W—I—K—D—F—S—T—;
40	Closed-loop turning canned cycle	G73P—Q—U—W—I—K—D—F—S—T—;
41	End face cutting-off canned cycle	G74X—Z—I—K—F—D—;
42	Outer surface/inner surface cutting-off canned cycle	G75X—Z—I—K—F—D—;

FS0-TC	FS0-TC tape format conversion	Function comparison
G34X—Z—F—K—;		A
G36X—; X axis G37X—; Z axis		D
G40		A
G41		A
G42		A
G50X—Z—;		A
G250		B
G251P—Q—;		B
G52X—Z—;		A
G53X—Z—;		A
G54 to G59		C
G65P—L—Argument;		A
G66P—L—Argument;		C
G67;		A
G68;		A
G69;		A
G70P—Q—;		A
G71U—R—; G71P—Q—U—W—D—F— S—T—;	G71P—Q—U—W—F— S—T—; It is impossible to specify a cutting allowance for roughing. (I,K) An attempt to specify it is ignored.	D (C)
G72U—R—; G72P—Q—U—W—F—S—T—;	G72P—Q—U—W—D— F—S—T—; It is impossible to specify a cutting allowance for roughing. (I,K) An attempt to specify it is ignored.	D (C)
G73U—R—; G73P—Q—U—W—F—S—T—;	G73P—Q—U—W—I— K—D—F—S—T—;	B (A)
G74R—; G74X—Z—P—Q—R—F—;	G74X—Z—I—K— F—D—;	B (A)
G75R—; G75X—Z—P—Q—R—F—;	G75X—Z—I—K— F—D—;	B (A)

5. PROGRAM FORMAT

Item	Function	FS15-TA
43	Multiple repetitive canned cycle for threading	G76X—Z—I—K—D—F—A— P—Q—;
44	Canned cycles for drilling	G8*X—Y—Z—R—Q—P—F—L—; L : Number of repeat
45	Outer surface/inner surface turning canned cycle	G90X—Z—I—F—;
46	Threading canned cycle	G92X—Z—I—F—Q—;
47	End face turning canned cycle	G94X—Z—K—F—;
48	Constant surface speed control	G96S—;
49	Constant surface speed control cancel	G97S—;
50	Feed per minute	G98
51	Feed pre revolution	G99
52	Sub program calling	M98P—L—;

FS0-TC	FS0-TC tape format conversion	Function comparison
G76P—Q—R—; G76X—Z—R—P—Q—F—;	G76X—Z—I—K—D—F— A—P—Q—; A threading method (P) can be selected only from P1 (single-edge cutting with constant cutting amount) and P2 (jigzag cutting with constant cut- ting amount). It is impossi- ble to specify an arbitrary angle for the tool tip (A). If an angle other than those that can be specified in Series 0 (0, 29, 30, 55, 60, or 80 degrees) is specified, alarm P/S 062 is gener- ated.	D (C)
G8*X (Z)—C—Z (X)—R—Q— P—F—K—; K : Number repeat	G8*X (Z)—C—Z (X)—R— Q—P—F—L—; L : Repetition count It is impossible to select an arbitrary drill axis.	D (C)
G90X—Z—R—F—;	G90X—Z—I—F—;	B (A)
G92X—Z—R—F—;	G92X—Z—I—F—Q—;	D (C)
G94X—Z—R—F—;	G94X—Z—K—F—;	B (A)
G96S—;		C
G97S—;		A
G98		A
G99		A
M98P—;	M98P—L—; If a number with more than three digits is specified in calling a subprogram, the lower three digits are re- garded as a subprogram number (address P). If the number of repetitive calls is not specified, it is as- sumed to be 1.	B (A)

5. PROGRAM FORMAT

FS0-MC/FS15-MA format comparison table

Function comparison

- A : Both models have the same functions and formats.
 B : Both models have the same functions, but different formats.
 C : Both models have the same formats, but different functions.
 D : The models have different functions and formats.

Item	Function	FS15-MA
1	Positioning	G00P—;
2	Linear interpolation	G01P—F—;
3	Circular interpolation	G17G02/G03X—Y—R—(I—J—)F—; G18G02/G03Z—X—R—(K—I—)F—; G19G02/G03Y— Z—R—(J—K—)F—;
4	Dwell	G98G04P—/X—/U—; (in time units) G99G04P—/X—/U—; (in rotation units)
5	Exact stop	G09
6	Data setting	G10L—P—R—;
7	Polar coordinate command cancel	G15
8	Polar coordinate command	G16
9	Plane selection	G17/G18/G19
10	Inch input	G20
11	Metric input	G21
12	Stored stroke check ON	G22X—Y—Z—I—J—K—;
13	Stored stroke check OFF	G23;
14	Reference position return check	G27P—;
15	1st reference position return	G28P—;
16	Return form reference position	G29P—;
17	nth reference position return	G30P2/3/4P—;
18	Skip cutting	G31P—;
19	Thread cutting	G33P—F—Q—; Q : Threading start shift angle G33X—Z—E—Q—; E : Number of screw threads per inch
20	Automatic tool offset	G37α—;
21	Corner arc	G39I/K/J—J/I/K—;
22	Cutter compensation cancel	G40
23	Cutter compensation (Left)	G41
24	Cutter compensation (Right)	G42
25	Tool length compensation (+)	G43
26	Tool length compensation (–)	G44
27	Tool offset increase	G45
28	Tool offset decrease	G46
29	Tool offset double increase	G47
30	Tool offset double decrease	G48

FS0-MC	FS0-MC tape format conversion	Function comparison
G00P—;		C
G01P—F—;		A
G17G02/G03X— Y—R—(I—J—)F—; G18G02/G03Z— X—R—(k—I—)F—; G19G02/G03Y— Z—R—(J—K—)F—;		C
G04P—/X—/U—; (in time units)		C
G09		A
G10L—P—R—;		A
G15		A
G16		A
G17/G18/G19		A
G20		C
G21		C
G22X—Y—Z—I—J—K—;		A
G23;		A
G27P—;		A
G28P—;		A
G29P—;		A
G30P2/3/4—;		A
G31P—;		C
G33P—F—;		D
G37α—;		A
G39X/Z/Y(I/K/J)—Y/X/Z(J/I/K)—;		B
G40		A
G41		A
G42		A
G43		A
G44		A
G45		A
G46		A
G47		A
G48		A

5. PROGRAM FORMAT

Item	Function	FS15-MA
31	Tool length compensation cancel	G49
32	Scaling cancel	G50
33	Scaling	G51X—Y—Z—P—;
34	Local coordinate system setting	G52P—;
35	Machine coordinate system select	(G90) G53P—;
36	Workpiece coordinate system select	G54 to G59
37	Single direction positioning	G60P—;
38	Exact stop mode	G61
39	Automatic corner override	G62
40	Tapping mode	G63
41	Cutting mode	G64
42	Macro calling	G65P—L—Argument;
43	Macro modal calling	G66P—L—Argument;
44	Macro modal calling cancel	G67;
45	Coordinate rotation	(G17/G18/G19) G68 α — β —R—;
46	Coordinate rotation cancel	G69;
47	Canned cycles for drilling	G**X—Y—Z—R—Q—P—F—L—; L : Number of repeat
48	Rigid tapping	M**S—; G84.2/3X—Y—Z—R—Q—P— F—L—; L : Number of repeat
49	Absolute command	G90
50	Increment command	G91
51	Coordinate system setting	(G90) G92P—;
52	Feed per minute	G94
53	Feed per revolution	G95
54	Constant surface speed control	G96S—;
55	Constant surface speed control cancel	G97S—;
56	Sub program calling	M98P—L—;

FS0-MC	FS0-MC tape format conversion	Function comparison
G49		A
G50		A
G51X—Y—Z—P—;		A
G52P—;		A
(G90) G53P—;		A
G54 to G59		C
G60P—;		C
G61		A
G62		A
G63		A
G64		A
G65P—L—Argument;		A
G66P—L—Argument;		C
G67;		A
(G17/G18/G19) G68 α — β —R—;		A
G69;		A
G**X—Y—Z—R—Q—P— F—K—; K : Number of repeat	G**X—Y—Z—R—Q— P—F—L—; L : Number of repeat	D (C)
M**S—; G84X—Y—Z—R—Q—P— F—K—; K : Number of repeat	M**S—; G84X—Y—Z—R—Q— P—F—L—; L : Number of repeat	B (A)
G90		A
G91		A
(G90) G92P—;		A
G94		A
G95		A
G96S—;		A
G97S—;		A
M98P—;	M98P—L—; If a number with more than three digits is specified in calling a subprogram, the lower three digits are re- garded as a subprogram number (address P). If the number of repetitive calls is not specified, it is as- sumed to be 1.	B (A)

6. CUSTOM MACRO

6.1 Custom Macro A

6.1.1 Types of variables

Type of variable	Variable number
Common variable	#100 – #149 #500 – #531
System variable	#1 – #99 #1000 – #5084

6.1.2 System variable

Variable number	Contents	Purpose	Series
#1000 – #1015	Corresponds to UI000 to UI015	Interface input signal	T/M
#1032	Unified input of UI000 to UI015		
#1100 – #1115	Corresponds to UO000 to UO015	Interface output signal	T/M
#1132	Unified output of UO000 to UO015		
#1133	Unified output of UO100 to UO131		
#2001 – #2032	Wear offset value (Offset No. 1 – 32)	X axis offset	T
#2701 – #2732	Geometry offset value (Offset No. 1 – 32)		
#2101 – #2132	Wear offset value (Offset No. 1 – 32)	Z axis offset	T
#2801 – #2832	Geometry offset value (Offset No. 1 – 32)		
#2201 – #2232	Wear offset value (Offset No. 1 – 32)	Tool nose radius compensation	T
#2901 – #2932	Geometry offset value (Offset No. 1 – 32)		
#2301 – #2332	Wear offset value (Offset No. 1 – 32)	Imaginary tool tip position	T
#2301 – #2332	Geometry offset value (Offset No. 1 – 32)		
#2401 – #2432	Wear offset value (Offset No. 1 – 32)	Y axis offset	T
#2451 – #2432	Geometry offset value (Offset No. 1 – 32)		
#2001 – #2200 (#1 – #99)	Tool compensation (Offset No. 1 – 99)	Tool compensation (Offset memory A)	M
#3011	Year, month, day	Clock	T/M
#3012	Hour, minute, second		
#3901	No. of parts machined	No. of parts	T/M
#3902	No. of parts machined		

Variable number	Contents	Purpose	Series
#4001 – #4022	G code (group 01 – 22)	Modal information	T/M
#4102	B code		
#4109	F code		
#4111	H code		
#4113	M code		
#4114	Sequence number		
#4115	Program number		
#4119	S code		
#4120	T code		
#5001 – #5004	1st axis block end position : 4th axis block end position	Block end position (Workpiece coordinate)	T/M
#5021 – #5024	1st axis current position : 4th axis current position	Machine coordinate	T/M
#5041 – #5044	1st axis current position : 4th axis current position	Workpiece coordinate	T/M
#5061 – #5064	1st axis skip signal position : 4th axis skip signal position	Skip signal position (Workpiece coordinate)	T/M
#5081 – #5084	1st axis tool offset value : 4th axis tool offset value	Tool offset value	T/M

6

6.1.3 Argument specification (M series)

Correspondence between addresses and variable numbers

Variable number (value)	Variable number (flag)	Address
#8004	#8104	I
#8005	#8105	J
#8006	#8106	K
#8009	#8109	F
#8010	#8110	G
#8011	#8111	H
#8013	#8113	M
#8014	#8114	N
#8016	#8116	P
#8017	#8117	Q
#8018	#8118	R
#8019	#8119	S
#8020	#8120	T
#8024	#8124	X
#8025	#8125	Y
#8026	#8126	Z

6. CUSTOM MACRO

Correspondence between G codes of the argument
specification and variable numbers

Variable number (value)	Variable number (flag)	G code group number	G codes of the argument specification
#8030	#8130	00	One shot and others
#8031	#8131	01	G00, G01, G02, G03
#8032	#8132	02	G17, G18, G19
#8033	#8133	03	G90, G91
#8035	#8135	05	G94
#8036	#8136	06	G20, G21
#8037	#8137	07	G40, G41, G42
#8038	#8138	08	G43, G44, G49
#8039	#8139	09	G73, G74, G76, G80 to G89
#8040	#8140	10	G98, G99
#8041	#8141	11	G50, G51
#8042	#8142	12	G66, G67
#8045	#8145	15	G61, G62, G63, G64
#8046	#8146	16	G68, G69

6.1.4 Operation instructions and branch instructions

G code	H code	Function	Definition
G65	H01	Definition, substitution	$\#i = \#j$
G65	H02	Addition	$\#i = \#j + \#k$
G65	H03	Subtraction	$\#i = \#j - \#k$
G65	H04	Product	$\#i = \#j \times \#k$
G65	H05	Division	$\#i = \#j \div \#k$
G65	H11	Logical sum	$\#i = \#j$. OR. $\#k$
G65	H12	Logical product	$\#i = \#j$. AND. $\#k$
G65	H13	Exclusive OR	$\#i = \#j$. XOR. $\#k$
G65	H21	Square root	$\#i = \sqrt{\#j}$
G65	H22	Absolute value	$\#i = \#j $
G65	H23	Remainder	$\#i = \#j - \text{trunc}(\#j / \#k) \times \#k$ (trunc : Discard fractions less than 1)
G65	H24	Conversion from BCD to binary	$\#i = \text{BIN}(\#j)$
G65	H25	Conversion from binary to BCD	$\#i = \text{BCD}(\#j)$
G65	H26	Combined multiplication/division	$\#i = (\#j \times \#k) \div \#k$
G65	H27	Combined square root 1	$\#i = \sqrt{\#J^2 + \#K^2}$
G65	H28	Combined square root 2	$\#i = \sqrt{\#J^2 - \#K^2}$
G65	H31	Sine	$\#i = \#j \cdot \text{SIN}(\#k)$
G65	H32	Cosine	$\#i = \#j \cdot \text{COS}(\#k)$
G65	H33	Tangent	$\#i = \#j \cdot \text{TAN}(\#k)$
G65	H34	Arctangent	$\#i = \text{ATAN}(\#j / \#k)$
G65	H80	Unconditional divergence	GOTOn
G65	H81	Conditional divergence 1	IF $\#j = \#k$, GOTOn
G65	H82	Conditional divergence 2	IF $\#j \neq \#k$, GOTOn
G65	H83	Conditional divergence 3	IF $\#j > \#k$, GOTOn
G65	H84	Conditional divergence 4	IF $\#j < \#k$, GOTOn
G65	H85	Conditional divergence 5	IF $\#j \geq \#k$, GOTOn
G65	H86	Conditional divergence 6	IF $\#j \leq \#k$, GOTOn
G65	H99	P/S alarm occurrence	P/S alarm number 500 +n occurrence

6. CUSTOM MACRO

6.1.5 Macro call

Name	Format	Program No.	Parameter No.	Remarks
Simple call	M98 (program number);			Refer to 6.1.3 for argument assignment.
Modal call (M series)	G66P (program number); L (repetition count) G67;			
Subprogram call by M code	Mxx;	9001 9002 9003	240 241 242	Displayed on program check screen but no MF nor M code is sent. Set an M code that calls a subprogram specified by the parameter.
	Max. 3 M codes from M006 – M255			
Subprogram call by T code	Tt;	9000	040#5	Calls subprogram 9000. T code t is stored in common variable #149 as an argument.

6.2 Custom Macro B

6.2.1 Types of variables

Type of variable	Variable number
Local variable	#1 – #33
Common variable	#100 – #149 #500 – #531
Additional common variable (NOTE 1)	#150 – #199 #532 – #999
System variable (NOTE 2)	#1000 – #19099

NOTE 1 Common variable #150 to #199 and #532 to #999 can be added.
Part program length reduces by 6.6 m.

NOTE 2 Details are shown 6.2.2.

6.2.2 System variable

Variable number	Contents	Purpose	Series
#1000 – #1015	Corresponds to UI000 to UI015	Interface input signal	T/M
#1032	Unified input of UI000 to UI015		
#1100 – #1115	Corresponds to UO000 to UO015	Interface output signal	T/M
#1132	Unified output of UO000 to UO015		
#1133	Unified output of UO100 to UO131		
#2001 – #2032	Wear offset value (Offset No. 1 – 32)	X axis offset	T
#2701 – #2732	Geometry offset value (Offset No. 1 – 32)		
#2101 – #2132	Wear offset value (Offset No. 1 – 32)	Z axis offset	T
#2801 – #2832	Geometry offset value (Offset No. 1 – 32)		
#2202 – #2232	Wear offset value (Offset No. 1 – 32)	Tool nose radius compensation	T
#2901 – #2932	Geometry offset value (Offset No. 1 – 32)		
#2301 – #2332	Wear offset value (Offset No. 1 – 32)	Imaginary tool tip position	T
#2301 – #2332	Geometry offset value (Offset No. 1 – 32)		
#2401 – #2432	Wear offset value (Offset No. 1 – 32)	Y axis offset	T
#2451 – #2482	Geometry offset value (Offset No. 1 – 32)		
#2001 – #2200	Tool compensation (Offset No. 1 – 200)	Tool compensation (Offset memory A)	M
#10001 – #10400	Tool compensation (Offset No. 1 – 400)		

6. CUSTOM MACRO

Variable number	Contents	Purpose	Series
#2001 – #2200	Wear offset value (Offset No. 1 – 200)	Tool compensa- tion (Offset memory B)	M
#2201 – #2400	Geometry offset value (Offset No. 1 – 200)		
#10001 – #10400	Wear offset value (Offset No. 1 – 400)		
#11001 – #11400	Geometry offset value (Offset No. 1 – 400)		
#3000		Alarm	T/M
#3001	Clock 1 (Unit: 1 ms)	Clock	T/M
#3002	Clock 2 (Unit: 1 hour)		
#3003		Control of single block stop, wait signal for FIN	T/M
#3004		Control of feed- hold, feedrate override, exact stop check	T/M
#3005		Setting	T/M
#3011	Year, month, day	Clock	T/M
#3012	Hour, minute, second		
#3901	No. of parts machined	No. of parts	T/M
#3902	No. of parts required		
#4001 – #4022	G code (group 01 – 22)	Model information	T/M
#4102	B code		
#4109	F code		
#4111	H code		
#4113	M code		
#4114	Sequence number		
#4115	Program number		
#4119	S code		
#4120	T code		
#5001 – #5004	1st axis block end posi- tion : 4th axis block end posi- tion	Block end position (Workpiece coordi- nate)	T/M
#5021 – #5024	1st axis current position : 4th axis current position	Machine coordi- nate	T/M
#5041 – #5044	1st axis current position : 4th axis current position	Workpiece coordi- nate	T/M
#5061 – #5064	1st axis skip signal posi- tion : 4th axis skip signal posi- tion	Skip signal posi- tion (Workpiece coordi- nate)	T/M
#5081 – #5084	1st axis tool offset value : 4th axis tool offset value	Tool offset value	T/M

Variable number	Contents	Purpose	Series
#5101 – #5104	1st axis servo position deviation : 4th axis servo position deviation	Servo position deviation	T/M
#2500 #2600 #2700 #2800	External workpiece zero point offset value	1st axis 2nd axis 3rd axis 4th axis	M
#2501 #2601 #2701 #2801	G54 workpiece zero point offset value	1st axis 2nd axis 3rd axis 4th axis	M
#2502 #2602 #2702 #2802	G55 workpiece zero point offset value	1st axis 2nd axis 3rd axis 4th axis	M
#2503 #2603 #2703 #2803	G56 workpiece zero point offset value	1st axis 2nd axis 3rd axis 4th axis	M
#2504 #2604 #2704 #2804	G57 workpiece zero point offset value	1st axis 2nd axis 3rd axis 4th axis	M
#2505 #2605 #2705 #2805	G58 workpiece zero point offset value	1st axis 2nd axis 3rd axis 4th axis	M
#2506 #2606 #2706 #2806	G59 workpiece zero point offset value	1st axis 2nd axis 3rd axis 4th axis	M
#2550 #2551 : #2556	External workpiece zero point offset value G54 (Workpiece coordinate system 1) : G59 (Workpiece coordinate system 6)	X axis	T
#2650 #2651 : #2656	External workpiece zero point offset value G54 (Workpiece coordinate system 1) : G59 (Workpiece coordinate system 6)	Z axis	T
#2750 #2751 : #2756	External workpiece zero point offset value G54 (Workpiece coordinate system 1) : G59 (Workpiece coordinate system 6)	C axis	T
#2850 #2851 : #2856	External workpiece zero point offset value G54 (Workpiece coordinate system 1) : G59 (Workpiece coordinate system 6)	Y axis	T

6. CUSTOM MACRO

Variable number	Contents	Purpose	Series
#7001 – #7004	G54 P1 workpiece zero point offset value	1st axis to 4th axis	M
#7021 – #7024	G54 P2 workpiece zero point offset value	1st axis to 4th axis	M
:	:	1st axis to 4th axis	
#7941 – #7944	G54 P48 workpiece zero point offset value	1st axis to 4th axis	M

6.2.3 Argument assignment I/II

**Correspondence
Table between Ar-
gument Assignment
I Addresses and
Macro Variables**

Argument Assignment I Addresses	Macro Variables
A	# 1
B	# 2
C	# 3
D	# 7
E	# 8
F	# 9
H	#11
I	# 4
J	# 5
K	# 6
M	#13
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

**Correspondence Table between Argument
Assignment II Addresses and Macro
Variables**

Argument Assignment II Addresses	Macro Variables
A	# 1
B	# 2
C	# 3
I ₁	# 4
J ₁	# 5
K ₁	# 6
I ₂	# 7
J ₂	# 8
K ₂	# 9
I ₃	#10
J ₃	#11
K ₃	#12
I ₄	#13
J ₄	#14
K ₄	#15
I ₅	#16
J ₅	#17
K ₅	#18
I ₆	#19
J ₆	#20
K ₆	#21

Argument Assignment II Addresses	Macro Variables
I ₇	#22
J ₇	#23
K ₇	#24
I ₈	#25
J ₈	#26
K ₈	#27
I ₉	#28
J ₉	#29
K ₉	#30
I ₁₀	#31
J ₁₀	#32
K ₁₀	#33

6.2.4 Arithmetic commands

Purpose	Expression	Contents
Definition and substitution of variables	#i=#j	Definition, substitution
Addition arithmetic	#i=#j+#k	Sum
	#i=#j-#k	Subtraction
	#i=#jOR#k	Logical sum (at every bit of 32 bits)
	#i=#jXOR#k	Exclusive OR (at every bit of 32 bits)
Multiplication arithmetic	#i=#j*#k	Product
	#i=#j/#k	Quotient
	#i=#jAND#k	Logical product (at every bit of 32 bits)
Functions	#i=SIN [#j]	Sine (degree unit)
	#i=COS [#j]	Cosine (degree unit)
	#i=TAN [#j]	Tangent (degree unit)
	#i=ATAN [#j]	Arctangent (degree unit)
	#i=SQRT [#j]/[#k]	Square root
	#i=ABS [#j]	Absolute value
	#i=BIN [#j]	Conversion from BCD to BIN
	#i=BCD [#j]	Conversion from BIN to BCD
	#i=ROUND [#j]	Rounding off
	#i=FIX [#j]	Discard fractions less than 1
Combination of arithmetic operations	#i=FUP [#j]	Add 1 for fractions less than 1
	—	The above arithmetic operations and functions can be combined. The order of priority in an arithmetic operation is function, multiplication arithmetic then addition arithmetic.

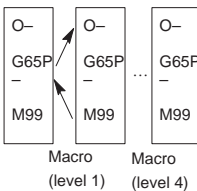
6

6.2.5 Control command

Purpose	Expression	Kind of operation
Conditional branch	IF [<conditional expression>] GOTO n Branch to sequence number n.	#j EQ #k (=)
		#j NE #k (≠)
		#j GT #k (>)
		#j LT #k (<)
		#jGE #k (≥)
		#j LE #k (≤)
Iteration	WHILE [<conditional expression>] DO m (m = 1, 2, 3) If omitted conditional expression, blocks from DO m to END m are executed eternally.	#j EQ #k (=)
		#j NE #k (≠)
		#j GT #k (>)
		#j LT #k (<)
		#jGE #k (≥)
		#j LE #k (≤)

6. CUSTOM MACRO

6.2.6 Macro call

Name	Format	Program No.	Parameter No.	Remarks
Simple call	G65P (program number) L (repetition count) <argument assignment>			Refer to 6.2.3 for argument assignment.
Modal call	G66P (program number); L (repetition count) <argument assignment>			
Macro call by G code	Gxx <argument assignment>	9010 : 9019	220 : 229	Refer to 6.2.3 for argument assignment. Set G or M code that calls a program specified in the parameter.
	Max. 10 G codes from G01–G64 and G68–G255			
Macro call by M code	Mxx <argument assignment>	9020 : 9029	230 : 239	
	Max. 10 M codes from M006 to M255			
Subprogram call by M code	Mxx;	9001 9002 9003	240 241 242	Displayed on program check screen but no MF nor M code is sent. Set an M code that calls a subprogram specified by the parameter.
	Max. 3 M codes from M006–M255			
Subprogram call by T code	Tt;	9000	045#5	Calls subprogram 9000. T code t is stored in common variable #149 as an argument.
Multiplex call	Main program 			Can be called up to 4 loops including simple call and modal call.

6.2.7 Command range

Item	Contents
Variables	Local variable: #1-#33 Common variable: #100-#149, #500-#531 Additional common variable: #100-#199, #500-#999 System variable: #1000-#19099
Value of variables	Maximum value $\pm 10^{47}$ Minimum value $\pm 10^{-29}$
Constant in expression	Maximum value ± 99999999 Minimum value ± 0.0000001 Decimal point possible
Arithmetic precision	Decimal 8 digits
Macro call duplex	Max. 4 loops
Iteration classification no.	1 to 3
Nesting	Max. 5 loops
Nesting of subprograms	Max. 4 loops (8 loops including macro calls)

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

7.1 CNC Internal Status Display

Self-diagnosis function displays

DGNOS number	Displays
000 to 022	States of signals supplied from the machine to the PMC. If the system has no PMC, only 016 to 022 are valid.
027	Signal indicating when the pulse coder or position coder makes one turn.
048 to 086	States of signals supplied from the PMC to the machine. If the system has no PMC, only 048 to 053 are valid.
100 to 147	States of signals supplied from the PMC to the CNC. If the system has no PMC, only 116 to 122 are valid.
148 to 199	States of signals supplied from the CNC to the PMC. If the system has no PMC, only 148 to 153 are valid.
200 to 249	Window data from the PMC to the CNC.
250 to 299	Window data from the CNC to the PMC.
700, 701	Internal state of the CNC. These are checked if a command is not responded with.
712	If a signal is turned off during automatic operation, this DGNOS number indicates the reason.
720 to 727	States of digital servo alarms.
760 to 767 770 to 777	States of serial pulse coder alarms.
800 to 807	Indicate a positional error for each axis.

CNC state display

If the CNC becomes inactive, but does not issue an alarm, press the DGNOS button to display the contents of DGNOS No. 700, 701, or 712. These DGNOS numbers show the state of the CNC.

0700		CSCT	CITL	COVZ	CINP	CDWL	CMTN	CFIN
------	--	------	------	------	------	------	------	------

When these bits are "1", they mean:

CFIN: The M, S, or T function is being executed.

CMTN: A tool movement command is being executed during automatic operation.

CDWL: A dwell command is being executed.

CINP: In position check is being made.

COVZ: The override is set to 0%.

CITL: The interlock signal is on.

CSCT: The spindle constant-speed reached signal is being awaited.

0701			CRST					
------	--	--	------	--	--	--	--	--

CRST: An emergency stop, external reset, reset & rewind signal, or a reset button on the MDI panel is on.

0702	STP	REST	EMS	RRW	RSTB	-	-	CSU
------	-----	------	-----	-----	------	---	---	-----

#7	#6	#5	#4	#3	#2	#1	#0	Cause
1	1	1	0	0	-	-	1	An emergency stop signal was supplied.
1	1	0	0	0	-	-	0	An external reset signal was supplied.
1	1	0	1	0	-	-	0	A reset & rewind signal was supplied.
1	1	0	0	1	-	-	0	The reset button was pressed.
1	0	0	0	0	-	-	1	A servo alarm occurred.
1	0	0	0	0	-	-	0	A feed hold signal was supplied, or the mode was switched.

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

The details about digital servo alarm No. 4n4 are indicated by diagnose numbers 720 to 727 in the order of the X-axis, Y-(Z-) axis, Z (C, PMC) axis, fourth (Y, PMC) axis, fifth to eighth axes.

DGNOS

No.

	#7	#6	#5	#4	#3	#2	#1	#0
720 to 727	OVFL	LV	OVC	HCAL	HVAL	DCAL	FBAL	OFAL

OFAL: An overflow alarm occurred.

FBAL: A wire breakage alarm occurred.

DCAL: A regenerative discharge alarm occurred.

HVAL: An overvoltage alarm occurred.

HCAL: An abnormal current alarm occurred.

OVC: An overcurrent alarm occurred.

LV: A low-voltage alarm occurred.

OVL: An overload alarm occurred.

The details of a serial pulse coder alarm No. 3n9 (n is an axis number, pulse coder alarm) are indicated by diagnose numbers 760 to 767 in the order of axis number.

	#7	#6	#5	#4	#3	#2	#1	#0
760 to 767	SRFLG	CSAL	BLAL	PHAL	SCAL	BZAL	CKAL	SPHAL

SRFLG: This is not an alarm bit. It is set to 1 when a serial pulse coder is connected, or to 0 when a conventional pulse coder is connected.

SPHAL: An abnormal software phase data alarm occurred. The serial pulse coder or feedback cable is probably defective. Replace it if necessary.

CKAL: A clock alarm occurred. The serial pulse coder is probably defective. Replace it if necessary.

BZAL: A battery zero alarm occurred (alarm No. 3n6). The pulse coder was supplied with power for the first time. Make sure that the battery is connected, and turn the power off and on again, then make a return to the zero point.

RCAL: An abnormal rotation speed count alarm occurred. The serial pulse coder is probably defective. Replace it if necessary.

PHAL: An abnormal phase data alarm occurred. The serial pulse coder or feedback cable is probably defective. Replace it if necessary.

BLAL: A low battery voltage alarm occurred (alarm No. 3n7). The battery voltage is too low. Replace the battery.

CSAL: A check sum alarm occurred. The serial pulse coder is probably defective. Replace it if necessary.

The details of a serial pulse coder alarm No. 3n9 (n is an axis number, communication alarm) are indicated by diagnose numbers 770 to 777 in the order of axis number.

	#7	#6	#5	#4	#3	#2	#1	#0
770 to 777	DTERR	CRCERR	STBERR					

DTERR: A data error occurred.

CRCERR: A CRC error occurred.

STBERR: A stop bit error occurred.

The probable causes of the above errors include a defective pulse coder, feedback cable, and feedback receive circuit.

Replace the pulse coder, feedback cable, or NC axis board.

Position error display (DGNOS numbers 800 to 807)

800 to 807	SVERRn (n is an axis number)
------------	------------------------------

The amount of position error for axis n is indicated.

NOTE For the 0-TT, the positional deviation for the first and second axes of the sub-tool post are indicated by 800 and 801 for the sub-tool post.

Machine position relative to the reference position (DGNOS numbers 820 to 825)

820 to 827	ABSMTr (n is an axis number.)
------------	-------------------------------

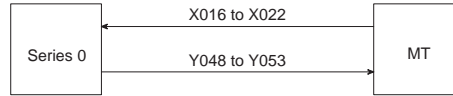
This machine position is also displayed on the position display page.

NOTE For the 0-TT, the machine position for the first and second axes of the sub-tool post are indicated by 820 and 821 for the sub-tool post.

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

7.2 I/O Signal Status Display between CNC and PMC or Machine Tool

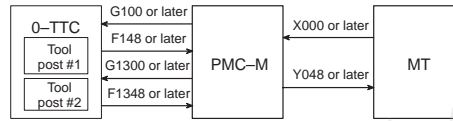
[For CNC without PMC]



[For CNC with PMC]



[For 0-TTC]



7.2.1 I/O signal list

Symbol in column of T series or M series
 ○ : Available ● : Available only 0-TTC
 ☆ : Available only 0-GSC/0-GCC Space : Unavailable

Symbol	Signal name	T	M	PMC address
*+EDCX, *-EDCX, *+EDCZ, *-EDCZ	External deceleration signal	○		G138.0, G138.3, G138.1, G138.4
*+LX, *+LY, *+LZ, *-LX, *-LY, *-LZ	Over travel limit signal		○	X020.0, X020.1, X020.2, X020.3, X020.4, X020.5
*+LZ	Over travel limit signal	○		X018.5
*+MITX, *+MITY, *+MITZ, *+MIT4, *-MITX, *-MITY, *-MITZ, *-MIT4	Interlock signal for each axis and direction		○	G142.0, G142.1, G142.2, G142.3, G142.4, G142.5, G142.6, G142.7
*-EDCX, *+EDCX, *-EDCY, *+EDCY, *-EDCZ, *+EDCZ	External deceleration signal		○	G138.3, G138.0, G138.4, G138.1, G138.5, G138.2
*ABSM	Manual absolute signal	○	○	G127.2
*AOV16, *AOV32, *AOV64, *AOV128	1% step override signal		○	G116.4, G116.5, G116.6, G117.6
*AOVR16, *AOVR32, *AOVR64, *AOVR128		○		G140.4, G140.5, G140.6, G140.7
*BECLP	B-axis clamp completion signal		○	G143.7
*BEUCP	B-axis unclamp completion signal		○	G143.6
*DECX, *DECY, *DECZ, *DEC4,	Deceleration signal for reference position return		○	X016.5, X017.5, X018.5, X019.5

Symbol	Signal name	T	M	PMC address
*DECX, *DECZ, *DEC3, *DEC4,	Deceleration signal for reference position return	○		X016.5, X017.5, X018.5, X019.5
*EAXSL	Control axis selection status signal (PMC axis control)	○	○	F188.7
*ESP, *ESP	Emergency stop signal	○	○	X021.4, G121.4
*ESPA, *ESPB	Emergency stop signal of spindle	○		G230.1, G234.1
*FLWU	Follow-up signal	○	○	G104.5
*ILK	Interlock signal		○	G117.0
*ITX, *ITY, *ITZ, *IT4	Interlock signal for each axis		○	G128.0, G128.1, G128.2, G128.3
*LDSP	Ladder display signal	○	○	G119.0
*OV1, *OV2, *OV4, *OV8	Override signal	○	○	G121.0, G121.1, G121.2, G121.3
*OV1E, *OV2E, *OV4E, *OV8E, *OV16E	Feedrate override signal (PMC axis control)	○	○	G147.0, G147.1, G147.2, G147.3, G147.4
*RILK	High-speed interlock signal		○	X008.5
*SCPF	Spindle clamp completion signal	○		G123.5
*SP	Automatic operation hold signal	○	○	G121.5
*SSTP	Spindle stop signal	○	○	G120.6
*SSTP1, *SSTP2, *SSTP3	Individual spindle stop signal	○		G145.3, G145.4, G145.5
*SUCPF	Spindle unclamp completion signal	○		G123.4
+3O, -3O	Jog feed axis select signal (software operator's panel signal)	○		F177.4, F177.5
+4O, -4O		○	○	F177.6, F177.7
+4, -4	Feed axis direction select signal	○	○	G119.2, G119.3
+LMX, +LMY, +LMZ, -LMX, -LMY, -LMZ	Software limit external setting		○	G129.0, G129.1, G129.2, G129.3, G129.4, G129.5
+MIT1, -MIT1, +MIT2, -MIT2	Interlock signal for each axis direction	○		X008.2, X008.3 X008.4, X008.5
+X, -X, +Z, -Z	Feed axis direction select signal	○		G116.2, G116.3, G117.2, G117.3
+X, -X, -Y, +Y, -Z, +Z			○	G116.2, G116.3, G117.3, G117.2, G118.3, G118.2
+XO, -XO, +YO, -YO, +ZO, -ZO	Jog feed axis select signal (software operator's panel signal)		○	F177.0, F177.1, F177.2, F177.3, F177.4, F177.5
+XO, -XO, +ZO, -ZO		○		F177.0, F177.1, F177.2, F177.3
4NG	4 axes disable signal		○	X004.7
AFL	Miscellaneous function lock signal	○	○	G103.7
AL	CNC alarm signal	○	○	F149.0
ALMA, ALMB	Spindle alarm signal		○	F281.0, F285.0

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Symbol	Signal name	T	M	PMC address
AR0 to AR15	Actual spindle speed signal	○		F158.0 to F159.7
ARSTA, ARSTB	Alarm reset signal	○	○	G230.0, G234.0
B0 to B31	2nd auxiliary function signal	○		F276.0 to F279.7
B11 to B38			○	F155.0 to F154.3
BAL	Battery alarm	○	○	F149.2
BAL1 to BAL4, BAL7, BAL8	Absolute pulse coder battery alarm	○		F156.0 to F156.5
BAL1 to BAL4, BAL7, BAL8			○	F159.0 to F159.5
BCLP	B axis clamp signal		○	F188.3
BDT1, BDT2 to BDT9	Optional block skip signal	○	○	G116.0, G141.0 to G141.7
BDTO	Optional block skip signal (software operator's panel signal)	○	○	F176.4
BF	B code strobe signal	○		F150.7
BF1, BF2	B code strobe signal		○	F150.7, F150.6
BFIN	2nd auxiliary function completion signal	○		G115.7
BFIN1, BFIN2	2nd auxiliary function completion signal		○	G115.7, G115.6
BGEACT	Background busy signal	○	○	F180.4
BOFF	Interference check invalid alarm signal	●		F180.6
BUCLP	B axis unclamp signal		○	F188.2
CDZ	Chamfering signal	○		G126.7
CFINA, CFINB	Spindle switch completion signal	○	○	F282.1, F286.1
CHPA, CHPB	Power line switch signal	○	○	F282.0, F286.0
CKGRP	Drawing signal	○	○	F164.5
COFF	Cs contour control change signal	○		G123.0
CON			○	G123.7
COSP	Spindle command signal	○		F180.5
CTH1A, CTH2A	Clutch/gear signal (serial spindle)	○	○	G229.3, G229.2
CUT	Cutting feed signal	○	○	F188.6
DEN	Distribution end signal	○	○	F149.3
DEN2	Passed point output signal	○		F149.6
DLK	Renewal disable signal of relative coordinate	○	○	G127.6
DMMC	DNC signal by data from MMC	○	○	G128.7
DNCI	DNC operation select signal	○	○	G127.5
DRN	Dry run signal	○		G118.7

Symbol	Signal name	T	M	PMC address
DRNE	Dry run signal (PMC axis control)	○	○	G147.7
DRNO	Dry run signal (software operator's panel signal)	○	○	F176.7
DST	Manual data input start signal	○	○	F150.5
EA0 to EA6	Address signal (for external data input)		○	G102.0 to G102.6
EAX1 to EAX8	Control axis select signal (PMC axis control)	○	○	G144.0 to G144.5
EBSYA, EBSYB	Command read completion signal (PMC axis control)	○	○	F270.7, F273.7
EBUFA, EBUFB	Command read signal (PMC axis control)	○	○	G210.7, G218.7
EC0A to EC7A, EC0B to EC7B	Axis control command signal (PMC axis control)	○	○	G211.0 to G211.7, G219.0 to G219.7
ECKZA, ECZKB	Following zero checking signal(PMC axis control)	○	○	F270.1, F273.1
ECLRA, ECLRB	Reset signal (PMC axis control)	○	○	G210.6, G218.6
ED0 to ED15	Data signal (for external data input)	○	○	G100.0 to G101.7
EDENA, EDENB	Auxiliary function executing signal (PMC axis control)	○	○	F270.3, F273.3
EF	External operation signal		○	F150.0
EFD	External operation signal for high-speed interface		○	F150.1
EFIN	External operation completion signal for high-speed interface		○	G115.1
EFINA, EFINB	Auxiliary function completion signal (PMC axis control)	○	○	G210.0, G218.0
EGENA, EGENB	Axis moving signal (PMC axis control)	○	○	F270.4, F273.4
EIALA, EIALB	Alarm signal (PMC axis control)	○	○	F270.2, F273.2
EID0A to EID31A, EID0B to EID31B	Axis control data signal (PMC axis control)	○	○	G214.0 to G217.7, G222.0 to G225.7
EIF0A to EIF15A, EIF0B to EIF15B	Axis control feedrate signal (PMC axis control)	○	○	G212.0 to G213.7, G220.0 to G221.7
EINPA, EINPB	In-position signal (PMC axis control)	○	○	F270.0, F273.0
EM11A to EM28A, EM11B to EM28B	Auxiliary function code signal (PMC axis control)	○	○	F272.0 to F272.7, F275.0 to F275.7
EMFA, EMFB	Auxiliary function strobe signal (PMC axis control)	○	○	F271.0, F274.0

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Symbol	Signal name	T	M	PMC address
EMSBKA, EMSBKB	Block stop disable signal (PMC axis control)	○	○	F211.0, G219.0
ENB	Spindle enable signal	○	○	F149.4
ENB2, ENB3		○	○	F149.4, F164.2, F164.3
ENBKY	External key input mode select signal	○	○	G134.0
EOTNA, EOTNB	Negative-direction overtravel signal (PMC axis control)	○	○	F270.6, F273.6
EOTPA, EOTPB	Positive-direction overtravel signal (PMC axis control)	○	○	F270.5, F273.5
EOV0	Override 0% signal (PMC axis control)	○	○	F188.5
EREND	Read completion signal (for external data input)	○	○	F160.0
ERS	External reset signal	○	○	G121.7
ESBKA, ESBKB	Block stop signal (PMC axis control)	○	○	G210.3, G218.3
ESEND	Search completion signal (for external data input)	○	○	F160.1
ESKIP	Skip signal (PMC axis control)		○	X008.6
ESOFA, ESOFB	Servo off signal (PMC axis control)	○	○	G210.4, G218.4
ESRSYC	Simple spindle synchronous control signal	○	○	G104.4
ESTB	Read signal (for external data input)	○	○	G102.7
ESTPA, ESTPB	Axis control temporary stop signal (PMC axis control)	○	○	G210.5, G218.5
EXLMZ	Stored stroke limit select signal	○	○	G129.6
EXRD	External read start signal	○	○	G134.1
EXSTP	External read/punch stop signal	○	○	G134.2
EXWT	External punch start signal	○	○	G134.3
F1D	F1-digit feed select signal		○	G140.7
FIN	M, S, T, B function completion signal	○	○	G120.3
FSCSL	Cs contour control change completion signal		○	F178.1
FSPPH	Serial spindle phase synchronous completion signal	○	○	F178.3
FSPSY	Serial spindle synchronous mode signal	○	○	F178.2

Symbol	Signal name	T	M	PMC address
FXST	Canned cycle start signal		○	F161.4
GOQSM	Offset write mode signal	○		G132.7
GR1, GR2	Gear select signal	○		G118.2, G118.3
GR1, GR2			○	G123.2, G123.3
GR10, GR20, GR30	Gear select signal		○	G152.0, G152.1, G152.2
GR21	2nd spindle gear select signal	○		G145.6
GR31	3rd spindle gear select signal	○		G145.7
GWLF	Automatic grinding wheel diameter compensation		○	F165.3
H4O	software operator's panel signal		○	F174.7
HIX, HIY, HIZ, HI4	Manual handle interruption axis select signal		○	G126.0, G126.1, G126.2, G126.3
HIX, HIZ, HI3, HI4		○		G126.0, G126.1, G126.2, G126.3
HX, HY, HZ, H4	Handle feed axis select signal		○	G116.7, G117.7, G118.7, G119.7
HX, HZ, H3, H4		○		G116.7, G117.7, G118.7, G119.7
HXO, HYO, HZO	Manual pulse generator feed axis select signal (software operator's panel signal)		○	F174.4, F174.5, F174.6
HXO, HZO, H3O, H4O		○		F174.4, F174.5, F174.6, F174.7
IGNVRY	V-READY check signal	○		G127.0
IGNVRY			○	G123.0
INDXA, INDXB	Spindle orientation stop position change signal		○	G231.0, G235.0
INFD	Infeed control cutting start signal		○	G237.6
INHKEY	Key input disable signal	○	○	F180.0
INP1 to INP4	In-position signal	○	○	F184.0 to F184.3
ITX, ITZ, IT3, IT4	Interlock signal for each axis	○		G128.0, G128.1, G128.2, G128.3
JOV1 to JOV8	Jog feedrate override signal		○	G104.0 to G104.3
JV10, JV20, JV40, JV80	Jog feedrate override signal (software operator's panel signal)	○		F175.0, F175.1, F175.2, F175.3
JV10 to JV80	Feedrate override signal (software operator's panel signal)		○	F175.0 to F175.3

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Symbol	Signal name	T	M	PMC address
K0, S, K1, U, K2, W, K3, R, K4, X, K5, Z, K6, F, K7, 0, K8, N, K9, G, KA, C, KALARM, KALTER, KCAN, KCUS↑, KCUS↓, KDELET, KDG NOS, KEOB	Key signal	○		G106.0, G106.1, G106.2, G106.3, G106.4, G106.5, G106.6, G106.7, G107.0, G107.1, G107.5, G109.5, G108.0, G110.7, G110.1, G110.0, G108.2, G109.4, G107.7
KFF, KFFE, KFGB, KFI., KFK%, KFL+, KFM., KFN), KFO (, KFRC, KFS=, KFT*, KFUV, KFWH, KFX Y, KFZJ, KH, I, K		○		G114.3, G114.2, G113.2, G114.0, G114.1, G114.7, G114.4, G113.1, G113.0, G113.3, G114.5, G114.6, G113.6, G113.7, G113.4, G113.5, G107.6
KINPUT, KINSRT, KM, -, KP, Q, KPAGE ↑, KPAGE ↓, KPARAM, KPOS, KPROGRAM, KRESET, KS1, KS2, KS3, KS4, KS5, KSL, KSR, KSTART, KT,, KXZ, KXZ		○		G108.3, G108.1, G107.2, G107.4, G110.3, G110.2, G109.3, G109.0, G109.1, G111.7, G112.6, G112.5, G112.4, G112.3, G112.2, G112.7, G112.1, G108.4, G107.3, G142.7, G110.4
K0, S, K1, H, K2, F, K3, R, K4, X, K5, Y, K6, Z, K7, 0, K8, N, K9, G, KALARM, KALTER, KB, K4, D, KCAN, KCUS↑, KCUS↓, KDELET, KDG NOS, KEOB, / , .		○		G106.0, G106.1, G106.2, G106.3, G106.4, G106.5, G106.6, G106.7, G107.0, G107.1, G109.5, G108.0, G107.5, G110.7, G110.1, G110.0, G108.2, G109.4, G107.7
KF4TH, KFF, KFF%, KFGE, KFI., KFJA, KFL+, KFM., KFN), KFO (, KFRC, KFS=, KFT*, KFXU, KFXW, KFYV, KH, J, K, KIN- PUT, KINSRT, KM, -, KOFSET, KP, Q, L, KPAGE↑, KPAGE↓, KPARAM, KPOS, KPROGRAM, KRESET, KS1, KS2, KS3, KS4, KS5, KSL, KSR, KSTART, KT,, KXY, KYZ, KZX		○		G113.7, G114.3, G114.2, G113.2, G114.0, G114.1, G114.7, G114.4, G113.1, G113.0, G113.3, G114.5, G114.6, G113.4, G113.6, G113.5, G107.6, G108.3, G108.1, G107.2, G109.2, G107.4, G110.3, G110.2, G109.3, G109.0, G109.1, G111.7, G112.6, G112.5, G112.4, G112.3, G112.2, G112.7, G112.1, G108.4, G107.3, G110.4, G110.6, G110.5
KEY	Program protect sig- nal	○	○	G122.3
KEYO	Program protect sig- nal (software opera- tor's panel signal)	○	○	F178.5

Symbol	Signal name	T	M	PMC address
KILPLUS	Position coder return direction select signal	○	○	G105.6
KOFSET	External key input	○		G109.2
LDT1A, LDT1B, LDT2A, LDT2B	Load detection signal	○	○	F281.4, F285.4, F281.5, F285.5
M00, M01, M02, M30	M decode signal		○	F154.7, F154.6, F154.5, F154.4
M11, M12, M14, M18, M21, M22, M24, M28, M31, M32, M34, M38	Miscellaneous function code signal	○	○	F151.0, F151.1, F151.2, F151.3, F151.4, F151.5, F151.6, F151.7, F157.0, F157.1, F157.2, F157.3
M211, M212, M214, M218, M221, M222, M224, M228, M231, M232, M234, M238	2nd M function code signal	○	○	F193.0, F193.1, F193.2, F193.3, F193.4, F193.5, F193.6, F193.7, F194.0, F194.1, F194.2, F194.3
M311, M312, M314, M318, M321, M322, M324, M328, M331, M332, M334, M338	3rd M function code signal	○	○	F194.4, F194.5, F194.6, F194.7, F195.0, F195.1, F195.2, F195.3, F195.4, F195.5, F195.6, F195.7
MA	CNC ready signal	○	○	F149.7
MCFNA, MCFNB	Power line switch completion signal	○	○	G230.3, G234.3
MD1, MD2, MD4	Mode select signal	○	○	G122.0, G122.1, G122.2
MD1O, MD2O, MD4O	Mode select signal (software operator's panel signal)	○	○	F174.0, F174.1, F174.2
MF	Miscellaneous function strobe signal	○		F150.0
MF2, MF3	2nd or 3rd M function strobe signal	○		F157.4, F157.5
MFIN	Miscellaneous function completion signal	○	○	G115.0
MFIN2	2nd M function completion signal	○	○	G134.4
MFIN3	3rd M function completion signal	○	○	G134.5
MINP	External program input start signal		○	G120.0
MIX, MIZ	Mirror image signal	○		G127.0, G127.1, G127.7
MLK	All-axis machine lock signal	○	○	G117.1
MLKO	Machine lock signal (software operator's panel signal)	○	○	F176.6
MMI1, MMI2	Mirror image check signal	○	○	F158.0, F158.1
MMI1, MMI2, MMI4			○	F158.0, F158.1, F158.3

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Symbol	Signal name	T	M	PMC address
MP1, MP2	Manual handle feed amount select signal	○		G118.0, G119.0
MP1, MP2			○	G120.0, G120.1
MP1O, MP2O	Select of magnification of manual pulse generator (software operator's panel signal)	○	○	F175.6, F175.7
MRDYA, MRDYB	Machine ready signal (serial spindle)	○	○	G229.7, G233.7
MVX, MVY, MV3, MV4	Axis moving signal	○		F148.4 to F148.7
MVX, MVY, MVZ, MV4			○	F148.4 to F148.7
NOWT	No-wait signal	●		G133.1
NOZAGC	Angular axis control	☆		G133.6
NOZAGC	Z axis offset moving signal		☆	G237.5
NRROA, NRROB	Short-distant movement command while changing the orientation stop position signal		○	G231.2, G235.2
OFN0, OFN1, OFN2, OFN3, OFN4	Tool offset number select signal of tool offset measurement direct input	○		G132.0, G132.1, G132.2, G132.3, G132.4
OP	Automatic operation signal	○	○	F148.7
ORARA, ORARB	Spindle orientation completion signal	○	○	F281.7, F285.7
ORCMA, ORCMB	Orientation command signal	○	○	G229.6, G233.6
OUT0 to OUT7	Software operator's panel general-purpose switch signal	○	○	F171.0 to F171.7
OV1O, OV2O, OV4O, OV8O	Feedrate override signal (software operator's panel signal)	○	○	F176.0, F176.1, F176.2, F176.3
OVC	Override cancel signal	○	○	G126.4
OVCE	PMC axis override cancel signal	○	○	G147.5
PC2SLC	2nd position coder select signal	○		G146.7
PCFNA, PCFNB	Output switching completion signal	○	○	F280.3, F282.3, F282.3, F286.3
PCHPA, PCHPB	Output switching signal	○	○	F280.2, F282.2, F282.2, F286.2
PK1M, PK2M, PK3M, PK4M, PK5M, PK6M	Synchronous control parking signal	○		G238.0, G238.1, G238.2, G238.3, G238.4, G238.5
PKESS1	1st spindle parking signal	○	○	G138.6
PKESS2	2nd spindle parking signal	○	○	G138.7
PLCRVON	Position coder return control signal	○	○	G105.7
PN1, PN2, PN4, PN8	Workpiece number search signal	○	○	G122.4, G122.5, G122.6, G122.7

Symbol	Signal name	T	M	PMC address
PRC	Position record signal	○		G103.6
PRGDPL	Program screen display signal	○	○	F180.1
PRTSF	Required parts count reached signal	○	○	F164.7
PSW01 to PSW10	Position switch signal	○	○	F190, F191.0, F191.1
PSYN	Polygon machining in-progress signal	○		F160.7
PX0 to PX7, PZ0 to PZ7	Position signal output	○		F165.0 to F165.7, F167.0 to F167.7
R01O to R12O	S12-bit code signal	○	○	F172.0 to F173.3
RCHA, RCHB	Power line status check signal	○	○	G230.7, G234.7
RGSPM, RGSP	Spindle rotation direction signal		○	F165.1, F165.0
RGTAP, RGTPN	Rigid tapping signal		○	G135.0, G123.1
RGTPN		○	○	G123.1
RGTSP1, RGTSP2	Rigid tapping spindle select signal	○	○	G061.4, G061.5
RLSOT	Software limit external setting		○	G129.7
RMTDI0 to RMTDI7	Remote buffer input signal	○	○	G239.0 to G239.7
RMTDO0 to RMTDO7	Remote buffer output signal		○	F290.0 to F290.7
RMTDO0 to RMTDO7		○		F289.0 to F289.7
RO1I to R12I, RO1I2 to R12I2, RO1I3 to R12I3	Motor speed command spindle input signal	○	○	G124.0 to G125.3, G106.0 to G107.3, G108.0 to G109.3
ROTA, ROTAB	Spindle rotation direction command signal while changing the orientation stop position	○	○	G231.1, G235.1
ROV1, ROV2	Rapid traverse override signal		○	G116.7, G117.7
ROV1D, ROV2D, ROV3D	Rapid traverse override B signal	○		G116.4, G116.5, G116.6
ROV1E, ROV2E	PMC axis rapid traverse override signal	○	○	G146.0, G146.1
ROV1O, ROV2O	Rapid traverse override signal (software operator's panel signal)	○	○	F175.4, F175.5
RPALM	Read/punch alarm signal	○	○	F180.3
RPBSY	Read/punch in-progress signal	○	○	F180.2
RRW	Rewind signal	○	○	G104.6
RSLA, RSLB	Output switch request signal	○	○	G230.6, G234.6
RSMAX	Serial spindle synchronous polygon signal	●		F189.7
RST	Reset signal	○	○	F149.1

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Symbol	Signal name	T	M	PMC address
RT	Manual rapid traverse select signal	○	○	G121.6
RTE	Manual rapid traverse select signal (PMC axis control)	○	○	G147.6
RTO	Jog rapid traverse select signal (software operator's panel signal)	○	○	F178.6
RWD	Rewinding signal	○	○	F164.6
S11, S12, S14, S18, S21, S22, S24, S28	Spindle function code signal	○	○	F152.0, F152.1, F152.2, F152.3, F152.4, F152.5, F152.6, F152.7
SA	Servo ready completion signal	○	○	F148.6
SAR	Spindle speed reached signal	○	○	G120.4
SARA, SARB	Speed reached signal	○	○	F281.3, F285.3
SARA, SARB		○	○	F281.3, F285.3
SBK	Single block signal	○	○	G116.1
SBKO	Single block signal (software operator's panel signal)	○	○	F176.5
SBRT	Spindle synchronous polygon mode signal for 0-TTC	●	○	G146.6
SCLP	Spindle clamp signal	○	○	F164.0
SDTA, SDTB	Speed detection signal	○	○	F281.2, F285.2
SDTA, SDTB		○	○	F281.2, F285.2
SF	Spindle function strobe signal	○	○	F150.2
SFIN	Spindle function completion signal	○	○	G115.2
SFRA, SFRB	Spindle CW command signal	○	○	G229.5, G233.5
SGN, SGN2, SGN3	Spindle motor command polarity select signal	○	○	G125.5, G107.5, G109.5
SHA00 to SHA11	1st spindle orientation external stop position command signal	○	○	G110.0 to G110.7, G111.0 to G111.3
SHB00 to SHB11	2nd spindle orientation external stop position command signal	○	○	G112.0 to G112.7, G113.0 to G113.3
SIGN	Spindle analog voltage control signal	○	○	G125.7
SIND, SIND2, SIND3	Spindle motor speed command signal	○	○	G125.7, G107.7, G109.7
SKIP	Skip signal	○	○	X008.7
SKIP2, SKIP3, SKIP4		○	○	X008.2, X008.3, X008.4
SLHZ0, SLHZ1	Z axis manual handle select signal	○	○	G133.0, G133.1

Symbol	Signal name	T	M	PMC address
SLPCA, SLPCB	Spindle return select signal	○		G1333.2, G1333.3
SLSPA, SLSPB	Spindle analog voltage output select signal	○		G133.2, G133.3
SMZ	Error detect signal	○		G126.6
SOCNA, SOCNB	Soft start /stop cancel signal	○	○	G230.4, G234.4
SOR	Spindle orientation signal	○	○	G120.5
SPA, SPB, SPC	Spindle speed override signal		○	G103.3, G103.4, G103.5
SPA, SPB, SPC, SPD		○		G103.3, G103.4, G103.5, G103.2
SPAL	Spindle fluctuation detection alarm signal	○		F154.0
SPDS1, SPDS2, SPDS3, SPDS4	Signal by axis moving speed		○	F189.0, F189.1, F189.2, F189.3
SPL	Automatic operation halt signal	○	○	F148.4
SPO	Feed hold signal (software operator's panel signal)	○	○	F178.7
SPPHS	Serial spindle phase synchronous mode signal	○	○	G146.3
SPSLA, SPSLB	Spindle select signal	○	○	G230.2, G234.2
SPSTP	Spindle stop completion signal	○		G123.6
SPSYC	Serial spindle synchronous mode signal	○	○	G146.2
SRN	Program restart signal	○	○	G103.0
SRNMV	Program restart under way signal	○	○	F188.4
SRVA, SRVB	Spindle CCW command signal	○	○	G229.4, G233.4
SSIN, SSIN2, SSIN3	Spindle motor command polarity select signal	○	○	G125.6, G107.6, G109.6
SSTA, SSTB	Speed zero detection signal	○		F281.1, F285.1
SSTA, SSTB			○	F281.1, F285.1
ST	Automatic operation start lamp signal	○	○	G120.2
STL	Automatic operation start signal	○	○	F148.5
STLK	Start lock signal	○		G120.1
STRD	Input and run simultaneous mode select signal		○	G140.5
STWD	Output and run simultaneous mode select signal		○	G140.6
SUCLP	Spindle unclamp completion signal	○		F164.1

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Symbol	Signal name	T	M	PMC address
SVFX, SVFY, SVFZ, SVF4	Servo off signal		○	G105.0, G105.1, G105.2, G105.3
SVFX, SVFZ, SVF3, SVF4		○		G105.0, G105.1, G105.2, G105.3
SWS1, SWS2, SWS3	Spindle select signal	○		G145.0, G145.1, G145.2
SYCAL	Spindle synchronous control alarm signal	○		F178.4
SYCAL	Synchronous error detection signal		○	F178.4
SYN1M, SYN2M, SYN3M, SYN4M, SYN5M, SYN6M	Synchronous control start signal	●		G237.0, G237.1, G237.2, G237.3, G237.4, G237.5
SYN1OM, SYN2OM, SYN3OM, SYN4OM, SYN5OM, SYN6OM	Axis recomposition signal	●		F189.0, F189.1, F189.2, F189.3, F189.4, F189.5
SYNAL	Servo axis synchronous alarm signal		○	F192.7
SYNC4	Simple synchronous axis select signal		○	G237.3
SYNCJ	Simple synchronous control signal		○	G133.6
SYNCX, SYNCZ, SYNC3, SYNC4	Simple synchronous axis select signal	○		G237.0 to G237.3
T11 to T28	Tool function code signal	○		F153.0 to F153.7
T11 to T48			○	F153.0 to F156.7
TAL	Interference alarm signal	●		F180.7
TAP	Tapping signal	○	○	F149.5
TF	Tool function strobe signal	○	○	F150.3
TFIN	Tool function completion signal	○	○	G115.3
THRD	Thread cutting in-progress signal	○		F188.3
TL01, TL02, TL04, TL08, TL16, TL32, TL64	Tool group number signal	○	○	G139.0, G139.1, G139.2, G139.3, G139.4, G139.5, G139.6
TLCH	Tool change signal	○	○	F188.0
TLCHB	Tool life reached signal		○	F192.2
TLCHE	All-tool life reached signal		○	F192.1
TLCHI	Individual tool change signal		○	F192.0
TLMA, TLMB	Torque limit signal	○	○	F281.6, F285.6
TLMA, TLMB			○	F281.6, F285.6
TLMHA, TLMHB	High-speed torque limit signal	○	○	G229.1, G233.1
TLMLA, TMLLB	Low speed torque limit signal	○	○	G229.0, G233.0
TLNW	New tool select signal	○	○	F188.1

Symbol	Signal name	T	M	PMC address
TLRST	Tool change reset signal	○	○	G139.7
TLRSTI	Individual tool change reset signal		○	G140.4
TLSKP	Tool skip signal	○	○	G140.0
TNFS	Tool offset number automatic select disable signal	○		G132.5
TORQ1, TORQ2, TORQ3, TORQ4, TORQ7, TORQ8	Torque limit reached signal	○		F170.0, F170.1, F170.2, F170.3, F170.4, F170.5
TRT2, TRT2PC	Tool post select signal	●		X018.3, G133.0
UI000 to UI015	Input signal for custom macro	○	○	G130.0 to G131.7
UINT	Interrupt signal for custom macro	○	○	G140.3
UO000 to UO015, UO100 to UO131	Output signal for custom macro	○	○	F162.0 to F163.7, F196.0 to F199.7
WATO	Waiting signal	●		F160.6
WNRQ	Window request signal	○	○	G200.0
WNANS	Window completion signal	○	○	F250.0
WNWRT	Window write signal	○	○	G200.1
WNERR	Window error signal	○	○	F250.1
WOQSM	Workpiece coordinate system shift value write mode select signal	○		G132.6
WOSET	Workpiece coordinate system shift value write signal	○		G133.7
XAE, YAE, ZAE	Measuring position reached signal		○	X008.0, X008.1, X008.2
XAE, ZAE	Measuring position reached signal	○		X008.0, X008.1
XAES	Automatic tool offset function signal	○		X040.0
ZAES	Automatic tool offset function signal	○		X040.1
ZNG	Z axis command cancel signal		○	G103.6
ZP2X, ZP2Y, ZP2Z, ZP24	2nd reference position return completion signal		○	F161.0, F161.1, F161.2, F161.3, F166.6, F166.7
ZP2X, ZP2Z, ZP23, ZP24		○		F161.0, F161.1, F161.2, F161.3, F166.6, F166.7
ZP3X, ZP3Y, ZP3Z, ZP34	3rd reference position return completion signal		○	F169.0, F169.1, F169.2, F169.3, F166.0, F166.1
ZP3X, ZP3Z, ZP33, ZP34		○		F169.0, F169.1, F169.2, F169.3, F166.0, F166.1

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Symbol	Signal name	T	M	PMC address
ZP4X, ZP4Y, ZP4Z, ZP44	4th reference position return completion signal		○	F169.4, F169.5, F169.6, F169.7, F166.2, F166.3
ZP4X, ZP4Z, ZP43, ZP44		○		F169.4, F169.5, F169.6, F169.7, F166.2, F166.3
ZPX, ZPY, ZPZ, ZP4	Reference position return completion signal		○	F148.0, F148.1, F148.2, F148.3, F166.4, F166.5
ZPX, ZPZ, ZP3, ZP4		○		F148.0, F148.1, F148.2, F148.3, F166.4, F166.5
ZRFX, ZRFY, ZRFZ, ZRF4	Reference position establishment signal		○	F168.0, F168.1, F168.2, F168.3
ZRFX, ZRFZ, ZRF3, ZRF4		○		F168.0, F168.1, F168.2, F168.3
ZRN	Manual reference position return select signal	○	○	G120.7
ZRNO	Mode select signal (software operator's panel signal)	○	○	F174.3

Address list (for M series)

7.2.2 Address list

Address list (for M series)

MT→PMC								
	#7	#6	#5	#4	#3	#2	#1	#0
X004	4NG							
X008	SKIP	ESKIP	*RILK			ZAE	YAE	XAE
X016			*DECX		*DEC5S	*DEC7		
X017			*DECY		*DEC6S	*DEC8		
X018			*DECZ					
X019			*DEC4					
X020			*- LZ	*- LY	*- LX	*+ LZ	*+ LY	*+ LX
X021				*ESP				

PMC→CNC								
	#7	#6	#5	#4	#3	#2	#1	#0
G100	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G101	ED15	ED14	ED13	ED12	ED11	ED10	ED09	ED08
G102	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G103	AFL	ZNG	SPC	SPB	SPA			SRN
G104		RRW	*FLWU	ESRSYC	JOV8	JOV4	JOV2	JOV1
G105			SVF8	SVF7	SVF4	SVFZ	SVFY	SVFX
G106	K7, 0	K6, Z	K5, Y	K4, X	K3, R	K2, F	K1, H	K0, S
G107	KEOB, I, #	KH, J, K	KB, K4, D	KP, Q, L	KT, .	KM, -	K9, G	K8, N
G108				KSTART	KINPUT	KDELET	KINSRT	KALTER
G109			KALARM	KDGNOS	KPARAM	KOFSET	KPROGRM	KPOS
G110	KCAN	KYZ	KZX	KXY	KPAGE↑	KPAGE↓	KCUS↑	KCUS↓

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

G111	KRESET						
G112	KSL	KS1	KS2	KS3	KS4	KS5	KSR
G113	KF4TH	KFXW	KFYV	KFXU	KFRC	KFGE	KFN)
G114	KFL+	KFT*	KFS=	KFM#	KFF	KFF@	KFJA
G115	BFIN1	BFIN2			TFIN	SFIN	EFIN
G116	HX/ROV1	AOV64	AOV32	AOV16	-X	+X	SBK
G117	HY/ROV2	AOV128			-Y	+Y	MLK
G118	HZ/DRN				-Z	+Z	
G119	H4				-4	+4	*LDSP
G120	ZRN	*SSTP	SOR	SAR	FIN	ST	MP2
G121	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2
G122	PN8	PN4	PN2	PN1	KEY	MD4	MD2
G123	CON	RTNT	MSPC	RTRCT	GR2	GR1	RGTPN
G124	RO8I	RO7I	RO6I	RO5I	RO4I	RO3I	RO2I
G125	SIND	SSIN	SGN		R12I	R11I	R10I
G126				OVC	HI4	HIZ	HIY
G127	MIR4	DLK	DNCI			*ABSM	MIRY
G128	DMMC				*IT4	*ITZ	*ITY
G129	RLSOT	EXLM2	-LMZ	-LMY	-LMX	+LMZ	+LMY
G130	UI7	UI6	UI5	UI4	UI3	UI2	UI1
G131	UI15	UI14	UI13	UI12	UI11	UI10	UI9
G132				BGEN	BGIALM	BGION	

Address list (for M series)

G133		SYNCJ	ROV4	ROV3			SLHZ1	SLHZ0
G134			MFIN3	MFIN2	EXWT	EXSTP	EXRD	ENBKY
G135								RGTAP
G136								
G137								
G138	PKESS2	PKESS1	*-EDCZ	*-EDCY	*-EDCX	*+EDCZ	*+EDCY	*+EDCX
G139	TLRST	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G140	F1D	STWD	STRD	TLRSTI	UNIT	TL256	TL128	TLSKP
G141	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G142	*-MIT4	*-MITZ	*-MITY	*-MITX	*+MIT4	*+MITZ	*+MITY	*+MITX
G143	*BECLP	*BEUCP		*JV16	*JV8	*JV4	*JV2	*JV1
G144			EAX8	EAX7	EAX4	EAX3	EAX2	EAX1
G145								
G146					SPPHS	SPSYC	ROV1E	ROV2E
G147	DRNE	RTE	OVCE	*OV16E	*OV8E	*OV4E	*OV2E	*OV1E
CNC→PMC								
	#7	#6	#5	#4	#3	#2	#1	#0
F148	OP	SA	STL	SPL	ZP4	ZPZ/EF	ZPY	ZPX
F149	MA	DEN2	TAP	ENB	DEN	BAL	RST	AL
F150	BF1	BF2	DST		TF	SF	EFD	MF
F151	M28	M24	M22	M21	M18	M14	M12	M11
F152	S28	S24	S22	S21	S18	S14/GR3O	S12/GR2C	S11/GR1O
F153	T28	T24	T22	T21	T18	T14	T12	T11

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

F154	M00	M01	M02	M30	B38	B34	B32	B31
F155	B28	B24	B22	B21	B18	B14	B12	B11
F156	T48	T44	T42	T41	T38	T34	T32	T31
F157			MF3	MF2	M38	M34	M32	M31
F158								
F159			BAL6	BAL5	BAL4	BAL3	BAL2	BAL1
F160							ESEND	EREND
F161				FXST	ZP24	ZP2Z	ZP2Y	ZP2X
F162	U07	U06	U05	U04	U03	U02	U01	U00
F163	U15	U14	U13	U12	U11	U10	U09	U08
F164	PRTSF	RWD	CKGRP					
F165	HOBSYN		MSPCF	RTRCTF	GWLF		RGSPM	RGSP
F166	ZP28	ZP27	ZP8	ZP7	ZP48	ZP47	ZP38	ZP37
F167								
F168	TORQL				ZRF4	ZRFZ	ZRFY	ZRFX
F169	ZP44	ZP4Z	ZP4Y	ZP4X	ZP34	ZP3Z	ZP3Y	ZP3X
F170								
F171	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F172	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F173					R12O	R11O	R10O	R09O
F174	H4O	HZO	HYO	HXO	ZRNO	MD4O	MD2O	MD1O
F175	MP2O	MP1O	ROV2O	ROV1O	JV8O	JV4O	JV2O	JV1O

Address list (for M series)

F176	DRNO	MLKO	SBKO	BDTO	OV8O	OV4O	OV2O	OV1O
F177	-4O	+4O	-ZO	+ZO	-YO	+YO	-XO	+XO
F178	SPO	RTO	KEYO	SYCAL	FSPPH	FSPSY	FSCSL	
F179	EDGN	EPARM	EVAR	EPRG	EWTO	ESTPIO	ERDIO	IOLNK
F180	PECK2			BGEACT	RPALM	RPBSY	PRGDPL	INHKY
F181					SRLNO3	SRLNO2	SRLNO1	SRLNO0
F182								
F183								
F184	MV4	MV3	MVZ	MVX	INP4	INP3	INPZ	INPX
F185	S28	S24	S22	S21	S18	S14	S12	S11
F186	S48	S44	S42	S41	S38	S34	S32	S31
F187					S58	S54	S52	S51
F188	*EAXSL	CUT	EOV0	SRNMV	BCLP	BUCLP	TLNW	TLCH
F189					SPDS4	SPDS3	SPDS2	SPDS1
F190								
F191								
F192	SYNAL	RTPT				TLCHB	TLCHE	TLCHI
F193	M228	M224	M222	M221	M218	M214	M212	M211
F194	M318	M314	M312	M311	M238	M234	M232	M231
F195	M338	M334	M332	M331	M328	M324	M322	M321
F196	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F197	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

F198	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F199	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124
PMC→CNC								
	#7	#6	#5	#4	#3	#2	#1	#0
G200				PMC window DI				
to								
G209				PMC window DI				
G210	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA			EFINA
G211	EC7A	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G212	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G213	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G214	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G215	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G216	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G217	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A
G218	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB			EFINB
G219	EC7B	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G220	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G221	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G222	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G223	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G224	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G225	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B
G226								

Address list (for M series)

G227								
G228								
G229	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G230	RCHA	RSLA		SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G231						NRROA	ROTA	INDXA
G232								
G233	MRDYB	ORCMB	SFRB	SRVB	CTH1B	CTH2B	TLMHB	TLMLB
G234	RCHB	RSLB		SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
G235						NRROB	ROTAB	INDXB
G236								
G237		INFD		SYNC4				
G238								
G239	RMTDI7	RMTDI6	RMTDI5	RMTDI4	RMTDI3	RMTDI2	RMTDI1	RMTDI0
G240				Operator's panel DI				
to								
G249				Operator's panel DI				
PMC→CNC								
	#7	#6	#5	#4	#3	#2	#1	#0
F250				PMC window DO				
to								
F269				PMC window DO				
F270	EBSYA	EOTNA	EOTPA	EGENA	EFENA	EIALA	ECKZA	EINPA
F271								EMFA
F272	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

F273	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECKZB	EINPB
F274								EMFB
F275	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F276								
F277								
F278								
F279								
F280								
F281	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F282					PCFNA	PCHPA	CFINA	CHPA
F283								
F284								
F285	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
F286					PCFNB	PCHPB	CFINB	CHOB
F287								
F288								
F289								
F290	RMTD07	RMTD06	RMTD05	RMTD04	RMTD03	RMTD02	RMTD01	RMTD00
F291				Operator's panel DO				
to								
F299				Operator's panel DO				

Address list (for T series)

Address list (for T series)

MT→PMC

	#7	#6	#5	#4	#3	#2	#1	#0
X008	SKIP		-MIT2	+MIT2	-MIT1	+MIT1	ZAE	XAE
	SKIP	SKIP4	SKIP3	SKIP2	ZAE	XAE		

☐ Lower stage for 0-GC

X016	*DEC3		*DECX		*DEC5S	*DEC7		
------	-------	--	-------	--	--------	-------	--	--

☐ By parameter setting

X017	*DEC4		*DECZ		*DEC6S	*DEC8		
------	-------	--	-------	--	--------	-------	--	--

☐ By parameter setting

X018	*ESPS		*+LZ		TRT2			
------	-------	--	------	--	------	--	--	--

X019	*DEC3		*DEC4					
------	-------	--	-------	--	--	--	--	--

☐ By parameter setting

X020								
------	--	--	--	--	--	--	--	--

X021			*ESP					
------	--	--	------	--	--	--	--	--

PMC→CNC

	#7	#6	#5	#4	#3	#2	#1	#0
G100	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0

G101	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
------	------	------	------	------	------	------	-----	-----

G102	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
------	------	-----	-----	-----	-----	-----	-----	-----

G103	AFL	PRC	SPC	SPB	SPA	SPD		SRN
------	-----	-----	-----	-----	-----	-----	--	-----

G104		RRW	*FLWU	ESRSYC				
------	--	-----	-------	--------	--	--	--	--

G105	PLCRVON	KILPLUS	SVF8	SVF7	SVF4	SVF3	SVFZ	SVFX
------	---------	---------	------	------	------	------	------	------

G106	K7, 0	K6, F	K5, Z	K4, X	K3, R	K2, W	K1, U	K0, S
	M2R08I	M2R07I	M2R06I	M2R05I	M2R04I	M2R03I	M2R02I	M2R01I

G107	KEOB	KH, I, K	KA, C	KP, Q	KT, .	KM, -	K9, G	K8, N
	M2SIND		M2SGN		M2R12I	M2R11I	M2R10I	M2R09I

G108				KSTART	KINPUT	KDELET	KINSRT	KALTER
	M3R08I	M3R07I	M3R06I	M3R05I	M3R04I	M3R03I	M3R02I	M3R01I

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

G109			KALARM	KDGNOS	KPARAM	KOFSET	KPROGRM	KPOS
	M3SIND	M3SSIN	M3SGN		M3R12I	M3R11I	M3R10I	M3R09I

G110	KCAN			KXZ	KPAGE↑	KPAGE↓	KCUS↑	KCUS↓
	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00

G111	KRESET							
					SHA11	SHA10	SHA09	SHA08

G112	KSL	KS1	KS2	KS3	KS4	KS5	KSR	
	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00

G113	KFWH	KFUV	KFZJ	KFXV	KFRC	KFGB	KFN	KFO(
					SHB11	SHB10	SHB09	SHB08

G114	KFL+	KFT*	KFS=	KFM#	KFF	KFFE	KFK@	KFI,
------	------	------	------	------	-----	------	------	------

G115	BFIN				TFIN	SFIN		MFIN
------	------	--	--	--	------	------	--	------

G116	HX/ROV1	ROV3D	ROV2D	ROV1D	-X	+X	SBK	BDT
------	---------	-------	-------	-------	----	----	-----	-----

G117	HZ/ROV2				-Z	+Z	MLK	MP1/MINP
------	---------	--	--	--	----	----	-----	----------

G118	DRN				GR2	GR1		MP2
	H3				-3	+3		MP2

☐ Lower stage by parameter setting

G119	H4				-4	+4		*LDSP
------	----	--	--	--	----	----	--	-------

G120	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIX
------	-----	-------	-----	-----	-----	----	------	-----

G121	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
------	-----	----	-----	------	------	------	------	------

G122	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
------	-----	-----	-----	-----	-----	-----	-----	-----

G123		SPSTP	*SCPF	*SUCPF			RGTPN	COFF
	DRN	SPSTP	*SCPF	*SUCPF	GR2	GR1		COFF

☐ Lower stage by parameter setting

G124	RO8I	RO7I	RO6I	RO5I	RO4I	RO3I	RO2I	RO1I
------	------	------	------	------	------	------	------	------

G125	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
------	------	------	-----	--	------	------	------	------

Address list (for T series)

G126	CDZ	SMZ	*OV16	OVC	HI4	HI3	HIZ	HIX
G127		DLK	DNCI			*ABSM	MIZ	IGNVRY
G128	DMMC				IT4	IT3	ITZ	ITX
G129		EXLM2						
G130	UI7	UI6	UI5	UI4	UI3	UI2	UI1	UI0
G131	UI15	UI14	UI13	UI12	UI11	UI10	UI9	UI8
G132	GOQSM	WOQSM	TNFS	OFN4	OFN3	OFN2	OFN1	OFN0
G133	WOSET	NOZAGC			SLSPB	SLSPA	NOWT	TRT2PC
G134			MFIN3	MFIN2	EXWT	EXSTP	EXRD	ENBKY
G135								
G136								
G137								
G138	PKESS2	PKESS1		*-EDCZ	*-EDCX		*+EDCZ	*+EDCX
G139	TLRST	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G140	AOVR128	AOVR64	AOVR32	AOVR16	UNIT			TLSKP
G141	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G142	KXZ			BGEN	BGIALM	BGION		IOLACK
G143								
G144			EAX8	EAX7	EAX4	EAX3	EAX2	EAX1
G145	GR31	GR21	*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G146	PC2SLC	SBRT			SPPHS	SPSYC	ROV2E	ROV1E
G147	DRNE	RTE	OVCE	*OV16E	*OV8E	*OV4E	*OV2E	*OV1E

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

CNC→PMC								
	#7	#6	#5	#4	#3	#2	#1	#0
F148	OP	SA	STL	SPL	ZP4	ZP3	ZPZ	ZPX
F149	MA	DEN2	TAP	ENB	DEN	BAL	RST	AL
F150	BF		DST		TF	SF		MF
F151	M28	M24	M22	M21	M18	M14	M12	M11
F152	S28	S24	S22	S21	S18	S14	S12	S11
F153	T28	T24	T22	T21	T18	T14	T12	T11
F154								SPAL
F156			BAL6	BAL5	BAL4	BAL3	BAL2	BAL1
F157			MF3	MF2	M38	M34	M32	M31
F158	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F159	AR15	AR14	AR13	AR12	AR11	AR10	AR9	AR8
F160	PSYN	WATO					ESEND	EREND
F161					ZP24	ZP23	ZP2Z	ZP2X
F162	UO7	UO6	UO5	UO4	UO3	UO2	UO1	UO0
F163	UO15	UO14	UO13	UO12	UO11	UO10	UO9	UO8
F164	PRTSF	RWD	CKGRP		ENB3	ENB2	SUCLP	SCLP
F165	PX7	PX6	PX5	PX4	PX3	PX2	PX1	PX0
F166	ZP28	ZP27	ZP8	ZP7	ZP48	ZP47	ZP38	ZP37
F167	PZ7	PZ6	PZ5	PZ4	PZ3	PZ2	PZ1	PZ0
F168					ZRF4	ZRF3	ZRFZ	ZRFX
F169	ZP44	ZP43	ZP4Z	ZP4X	ZP34	ZP33	ZP3Z	ZP3X

Address list (for T series)

F170			TORQ8	TORQ7	TORQ4	TORQ3	TORQ2	TORQ1
F171	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F172	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F173					R12O	R11O	R10O	R09O
F174	H4O	H3O	HZO	HXO	ZRNO	MD4O	MD2O	MD1O
F175	MP2O	MP1O	ROV2O	ROV1O	JV8O	JV4O	JV2O	JV1O
F176	DRNO	MLKO	SBKO	BDO	OV8O	OV4O	OV2O	OV1O
F177	-4O	+4 O	-3O	+3O	-ZO	+Z O	-XO	+X O
F178	SPO	RTO	KEYO	SYCAL	FSPPH	FSPSY	FSCSL	
F179	EDGN	EPARM	EVAR	EPRG	EWIO	ESTPIO	ERDIO	IOLNK
F180	TAL	BOFF	COSP	BGEACT	RPALM	RPBSY	PRGDPL	INHXY
F181					SRLNO3	SRLNO2	SRLNO1	SRLNO0
F182								
F183								
F184	MV4	MV3	MVZ	MVX	INP4	INP3	INPZ	INPX
F185	S28	S24	S22	S21	S18	S14	S12	S11
F186	S48	S44	S42	S41	S38	S34	S32	S31
F187					S58	S54	S52	S51
F188	*EAXSL	CUT	EOV0	SRNMV	THRD		TLNW	TLCH
F189	RSMAX		SYN6OM	SYN5OM	SYN4OM	SYN3OM	SYN2OM	SYN1OM
F190								
F191								

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

F192								
F193	M228	M224	M222	M221	M218	M214	M212	M211
F194	M318	M314	M312	M311	M238	M234	M232	M231
F195	M338	M334	M332	M331	M328	M324	M322	M321
F196	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F197	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F198	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F199	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124
PMC→CMC								
	#7	#6	#5	#4	#3	#2	#1	#0
G200				PMC window DI				
to								
G209				PMC window DI				
G210	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA		EFINA	
G211	EC7A	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G212	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G213	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G214	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G215	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G216	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G217	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A
G218	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB			EFINB
G219	EC7B	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G220	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B

Address list (for T series)

G221	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G222	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G223	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G224	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G225	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B
G226								
G227								
G228								
G229	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G230	RCHA	RSLA		SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G231								
G232								
G233	MRDYB	ORCMB	SFRB	ERVB	CTH1B	CTH2B	TLMHB	TLMLB
G234	PCHB	RSLB		SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
G235								
G236								
G237			SYN6M	SYN5M	SYN4M	SYN3M	SYN2M	SYN1M
G238			PK6M	PK5M	PK4M	PK3M	PK2M	PK1M
G239	RMTD17	RMTD16	RMTD15	RMTD14	RMTD13	RMTD12	RMTD11	RMTD10
G240			Operator's panel DI					
to								
G249			Operator's panel DI					

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

CMC→PMC								
	#7	#6	#5	#4	#3	#2	#1	#0
F250				PMC window DO				
to								
F269				PMC window DO				
F270	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECKZA	EINPA
F271	EMFA							
F272	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F273	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECZKB	EINPB
F274								EMFB
F275	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F276	B7	B6	B5	B4	B3	B2	B1	B0
F277	B15	B14	B13	B12	B11	B10	B9	B8
F278	B23	B22	B21	B20	B19	B18	B17	B16
F279	B31	B30	B29	B28	B27	B26	B25	B24
F280	SBSY	SZP	SALR	SGEN	PCFNA	PCHPA	CFINA	CHPA
F281	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F282					PCFNA	PCHPA	CFINA	CHPA
F283								
F284								
F285	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
F286					PCFNB	PCHPB	CFINB	CHPB
F287								
F288								

Address list (for T series)

F289	RMTD07	RMTD06	RMTD05	RMTD04	RMTD03	RMTD02	RMTD01	RMTD00
F290			Operator's panel DO					
to								
F299			Operator's panel DO					

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Address list (for 0-TTC)

- Interface between CNC and PMC or machine tool

Diagnostic number and PMC address of 0-TTC machine interface signal

Diagnostic No.	PMC address	Direction of signal	Selection of tool post
000 to 040	X000 to X040	Machine tool → CNC	Tool post 1 and tool post 2
048 to 086	Y048 to Y086	PMC → Machine tool	Tool post 1 and tool post 2
100 to 147	G100 to G147	PMC → CNC	Tool post 1
148 to 199	F148 to F199	CNC → PMC	Tool post 1
200 to 249	G200 to G249	PMC → CNC	Tool post 1
250 to 299	F250 to F299	CNC → PMC	Tool post 1
100 to 147	G1300 to G1347	PMC → CNC	Tool post 2
148 to 187	F1348 to F1387	CNC → PMC	Tool post 2
200 to 209	G1400 to G1409	PMC → CNC	Tool post 2
250 to 279	F1450 to F1479	CNC → PMC	Tool post 2

Machine interface signal (MT → CNC) (for tool post 1, 2)

		#7	#6	#5	#4	#3	#2	#1	#0
X000	000								
		M18-36	M18-21	M18-5	M18-35	M18-20	M18-34	M18-19	M18-33
X002	002								
		M18-24	M18-8	M18-38	M18-23	M18-7	M18-37	M18-22	M18-6
X004	004								
		M18-11	M18-41	M18-26	M18-10	M18-40	M18-25	M18-9	M18-39
X006	006								
		M18-45	M18-14	M18-44	M18-13	M18-43	M18-12	M18-42	M18-27
X008	008	SKIPM		-MIT2M	+MIT2M	-MIT1M	+MIT1M	ZAEM	XAEM
		M18-49	M18-18	M18-48	M18-17	M18-47	M18-16	M18-46	M18-15

Address list (for 0-TTC)

X010	010					/	/	/	/
		M20-11	M20-41	M20-26	M20-10				
X012	012								
		M20-45	M20-14	M20-44	M20-13	M20-43	M20-12	M20-42	M20-27
X014	014								
		M20-49	M20-18	M20-48	M20-17	M20-47	M20-16	M20-46	M20-15
X016	016	*DEC3	/	*DECXM	/	*DECXS			
		M1-6		M1-38		M1-20	M1-21	M1-11	M1-12
X017	017	*DEC4	/	*DECZM	/	*DECZS			
		M1-7		M1-39		M1-22	M1-23	M1-9	M1-10
X018	018	*ESPS	/	*+LZM	/	TRT2		/	/
		M1-8		M1-40		M1-24	M1-25		
X019	019		/		/			/	/
		M20-40		M20-25		M20-9	M20-39		
X020	020								
		M1-13	M1-37	M1-5	M1-14	M1-15	M1-16	M1-17	M1-18
X021	021				*ESPM				
		M1-41	M1-26	M1-27	M1-19	M1-33	M1-34	M1-35	M1-36
X022	022								
		M1-42	M1-43	M1-44	M1-45	M1-46	M1-47	M1-48	M1-49
X040	040	SKIPS	*+LZS	-MIT2S	+MIT2S	-MIT1S	+MIT1S	ZAES	XAES
		M28-11	M28-17	M28-10	M28-16	M28-9	M28-15	M28-8	M28-14

NOTE An alphabet at the tail of a signal name indicates as follows :

□□ . . □M ; Signal for tool post 1

□□ . . □S ; Signal for tool post 2

When the signals are discriminated per page between tool post 1 and tool post 2, the alphabet at the tail is omitted.

Machine interface signal (PMC → MT) (for tool post 1, 2)

		#7	#6	#5	#4	#3	#2	#1	#0
Y048	048					/			
		M2-5	M2-6	M2-7	M2-8		M2-27	M2-26	M2-25
Y049	049		/	/			/		
		M2-9		M2-41	M2-22		M2-23	M2-24	

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Y050	050	/	/		/			/	
		M2-10			M2-20		M2-19	M2-21	
Y051	051								
		M2-33	M2-34	M2-35	M2-36	M2-37	M2-38	M2-39	M2-40
Y052	052								
		M2-11	M2-12	M2-13	M2-14	M2-15	M2-16	M2-17	M2-18
Y053	053								
		M2-42	M2-43	M2-44	M2-45	M2-46	M2-47	M2-48	M2-49
Y080	080								
		M19-8	M19-7	M19-6	M19-5	M19-4	M19-3	M19-2	M19-1
Y082	082								
		M19-16	M19-15	M19-14	M19-13	M19-12	M19-11	M19-10	M19-9
Y084	084								
		M20-36	M20-21	M20-5	M20-35	M20-20	M20-34	M20-19	M20-33
Y086	086								
		M20-24	M20-8	M20-38	M20-23	M20-7	M20-37	M20-22	M20-6

Machine interface signal (PMC → CNC) (for tool post 1)

		#7	#6	#5	#4	#3	#2	#1	#0
G100	100	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G101	101	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G102	102	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G103	103	AFL	PRC	SPC	SPB	SPA	SPD		SRN
G104	104		RRW	*FLWU					
G105	105					SVF4	SVF3	SVFZ	SVFX
G106	106	K7, 0	K6, F	K5, Z	K4, X	K3, R	K2, W	K1, U	K0, S
G107	107	KEOB	KH, I, K	KA, C	KP, Q	KT, .	KM, -	K9, G	K8, N
G108	108				KSTART	KINOUT	KDELET	KINSRT	KALTER
G109	109			KALARM	KDGNOS	KPARAM	KOFSET	KPRGRM	KPOS

Address list (for 0-TTC)

G110	110	KCAN			KXZ	KPAGE↑	KPAGE↓	KCUS↑	KCUS↓
G111	111	KRESET							
G112	112	KSL	KS1	KS2	KS3	KS4	KS5	KSR	
G113	113	KFWH	KFUV	KFZJ	KFXV	KFRC	KFGB	KFN)	KFO(
G114	114	KFL+	*KFT	KFS=	KFM#	KFF	KFE	KFK@	KFI,
G115	115	BFIN				TFIN	SFIN		MFIN
G116	116	HX/ROV1	ROV3D	ROV2D	ROV1D	-X	+X	SBK	BDT
G117	117	HZ/ROV2				-Z	+Z	MLK	MP1/MINP
G118	118	DRN				GR2	GR1		MP2
		H3				-3	+3		MP2
☐ Lower stage is selected when parameter No. 0031#5 is 1									
G119	119	H4				-4	+4		*LDSP
G120	120	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIX
G121	121	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
G122	122	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
G123	123		STSP	*SCPF	*SUCPF				COFF
		DRN	SPSTP	*SCPF	*SUCPF	GR2	GR1		COFF
☐ Lower stage is selected when parameter No. 0031#5 is 1									
G124	124	R081	R071	R061	R051	R041	R031	R021	R011
G125	125	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G126	126	CDZ	SMZ	*OV16	OVC	HI4	HI3	HIZ	HIX
G127	127			DNCI			*ABSM	MIZ	IGNVRY
G128	128					IT4	IT3	ITZ	ITX
G129	129								
G130	130	UI7	UI6	UI5	UI4	UI3	UI2	UI1	UI0

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

G131	131	UI15	UI14	UI13	UI12	UI11	UI10	UI9	UI8
G132	132	GOQSM	GOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0
G133	133	WOSET				SLSPB	SLSPA	NOWT	TRT2PC
G134	134			MFIN3	MFIN2	EXWT	EXSTP	EXRD	ENBKY
G135	135								
		For custom macro							
G136	136								
		For custom macro							
G137	137								
		For custom macro							
G138	138				—*EDCZ	*—EDCX		*+EDCZ	*+EDCX
G139	139								
G140	140					VINT			
G141	141	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G142	142								
G143	143								
G144	144					EAX4	EAX3	EAX2	EAX1
G145	145								
G146	146								
G147	147								
Machine interface signal (CNC → PMC) (for tool post 1)									
		#7	#6	#5	#4	#3	#2	#1	#0
F148	148	OP	SA	STL	SPL	ZP4	ZP3	ZPZ	ZPX
F149	149	MA		TAP	ENB	DEN	BAL	RST	AL
F150	150	BF		DST		TF	SF		MF
F151	151	M28	M24	M22	M21	M18	M14	M12	M11

Address list (for 0–TTC)

F152	152	S28	S24	S22	S21	S18	S14	S12	S11
F153	153	T28	T24	T22	T21	T18	T14	T12	T11
F154	154								SPAL
F155	155								
F156	156								
F157	157			MF3	MF2	M38	M34	M32	M31
F158	158	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F159	159	AR15	AR14	AR13	AR12	AR11	AR10	AR9	AR8
F160	160	PSYN	WATO					ESEND	EREND
F161	161					ZP24	ZP23	ZP2Z	ZP2X
F162	162	UO7	UO6	UO5	UO4	UO3	UO2	UO1	UO0
F163	163	UO15	UO14	UO13	UO12	UO11	UO10	UO9	UO8
F164	164	PRTSF	RWD					SUCLP	SCLP
F165	165	PX7	PX6	PX5	PX4	PX3	PX2	PX1	PX0
F166	166								
F167	167	PZ7	PZ6	PZ5	PZ4	PZ3	PZ2	PZ1	PZ0
F168	168								
F169	169								
F170	170								
F171	171	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F172	172	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F173	173					R12O	R11O	R10O	R09O
F174	174	H4O	H3O	H2O	H1O	ZRNO	MD4O	MD2O	MD1O

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

F175	175	MP2O	MP1O	ROV2O	ROV1O	JV8O	JV4O	JV2O	JV1O
F176	176	DRNO	MLKO	SBKO	BDTO	OV8O	OV4O	OV2O	OV1O
F177	177	-4O	+4O	-3O	+3O	-ZO	+ZO	-XO	+X O
F178	178	SPO	RTO	KEYO					
F179	179								
F180	180	TAL	BOFF	COSP	BGEACT	RPALM	RPBSY	PRGDPL	INHKY
F181	181								
F182	182								
		For custom macro							
F183	183								
F184	184								
F185	185								
F186	186								
F187	187								
F188	188								
F189	189								
F190	190								
F191	191								
F192	192								
F193	193	M228	M224	M222	M221	M218	M214	M212	M211
F194	194	M318	M314	M312	M311	M238	M234	M232	M231
F195	195	M338	M334	M332	M331	M328	M324	M322	M321
F196	196	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100

Address list (for 0-TTC)

F197	197	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F198	198	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F199	199	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124

Machine interface signal (PMC → CNC) (for tool post 1)

		#7	#6	#5	#4	#3	#2	#1	#0
G200	200				PMC window DI				
G201	201				PMC window DI				
G202	202				PMC window DI				
									[Kind of data]
G203	203				PMC window DI				
									[No. of words of data]
G204	204				PMC window DI				
									[Axis number]
G205	205				PMC window DI				
									[0 0]
G206	206				PMC window DI				
G207	207				PMC window DI				
G208	208				PMC window DI				
G209	209				PMC window DI				
G210	210	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA			EFINA
G211	211	EC7A	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G212	212	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G213	213	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G214	214	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G215	215	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G216	216	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

G217	217	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A
G218	218	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB			EFINB
G219	219	EC7B	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G220	220	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G221	221	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G222	222	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G223	223	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G224	224	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G225	225	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B
G226	226								
G227	227								
G228	228								
G229	229								
G230	230								
G231	231								
G232	232								
G233	233								
G234	234								
G235	235								
G236	236								
G237	237								
G238	238								
G239	239								

Address list (for 0–TTC)

G240	240				Operator's panel DI			
G241	241				Operator's panel DI			
G242	242				Operator's panel DI			
G243	243				Operator's panel DI			
G244	244				Operator's panel DI			
G245	245				Operator's panel DI			
G246	246				Operator's panel DI			
G247	247				Operator's panel DI			
G248	248				Operator's panel DI			
G249	249				Operator's panel DI			

Machine interface signal (CNC → PMC) (for tool post 1)

F250	250				PMC window DO			
F251	251				PMC window DO			
F252	252				PMC window DO			
								[Transferred data]
F253	253				PMC window DO			
								[Transferred data]
F254	254				PMC window DO			
								[Transferred data]
F255	255				PMC window DO			
								[Transferred data]
F256	256				PMC window DO			
								[Transferred data]
F257	257				PMC window DO			
								[Transferred data]
F258	258				PMC window DO			
								[Transferred data]

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

F259	259				PMC window DO				
									[Transferred data]
F260	260				PMC window DO				
									[Transferred data]
F261	261				PMC window DO				
									[Transferred data]
F262	262				PMC window DO				
									[Transferred data]
F263	263				PMC window DO				
									[Transferred data]
F264	264				PMC window DO				
									[Transferred data]
F265	265				PMC window DO				
									[Transferred data]
F266	266				PMC window DO				
									[Transferred data]
F267	267				PMC window DO				
									[Transferred data]
F268	268				PMC window DO				
F269	269				PMC window DO				
F270	270	EBSYA			EGENA	EDENA	EIALA	ECXZA	EINPA
F271	271								EMFA
F272	272	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F273	273	EBSYB			EGEND	EDENB	EIALB	ECKZB	EINPB
F274	274								EMFB
F275	275	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F276	276	B7	B6	B5	B4	B3	B2	B1	B0
F277	277	B15	B14	B13	B12	B11	B10	B9	B8

Address list (for 0–TTC)

F278	278	B23	B22	B21	B20	B19	B18	B17	B16
F279	279	B31	B30	B29	B28	B27	B26	B25	B24
F280	280								
F281	281								
F282	282								
F283	283								
F284	284								
F285	285								
F286	286								
F287	287								
F288	288								
F289	289								
F290	290				Operator's panel	DO			
F291	291				Operator's panel	DO			
F292	292				Operator's panel	DO			
F293	293				Operator's panel	DO			
F294	294				Operator's panel	DO			
F295	295				Operator's panel	DO			
F296	296				Operator's panel	DO			
F297	297				Operator's panel	DO			
F298	298				Operator's panel	DO			
F299	299				Operator's panel	DO			

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Machine interface signal (PMC → CNC) (for tool post 2)

		#7	#6	#5	#4	#3	#2	#1	#0
G1300	100	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G1301	101	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G1302	102	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G1303	103	AFL	PRC	SPC	SPB	SPA	SPD		SPN
G1304	104		RRW	*FLWU					
G1305	105							SVFZ	SVFX
G1306	106	K7, 0	K6, F	K5, Z	K4, X	K3, R	K2, W	K1, U	K0, S
G1307	107	KEOB	KH, I, K	KA, C	KP, Q	KT, .	KM, -	K9, G	K8, N
G1308	108				KSTART	KINOUT	KDELET	KINSRT	KALTER
G1309	109			KALARM	KDGNOS	KPARAM	KOFSET	KPRGRM	KPOS
G1310	110	KCAN			KXZ	KPAGE↑	KPAGE↓	KCUS↑	KCUS↓
G1311	111	KRESET							
G1312	112	KSL	KS1	KS2	KS3	KS4	KS5	KSR	
G1313	113	KFWH	KFUV	KFZJ	KFXV	KFRC	KFGB	KFN)	KFO(
G1314	114	KFL+	KFT*	KFS=	KFM#	KFF	KFE	KFK@	KFI,
G1315	115	BFIN				TFIN	SFIN		MFIN
G1316	116	HX/ROV1	ROV3D	ROV2D	ROV1D	-X	+X	SBK	BDT
G1317	117	HZ/ROV2				-Z	+Z	MLK	MP1/MINP
G1318	118	DRN				GR2	GR1		MP2
G1319	119								
G1320	120	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIX
G1321	121	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1

Address list (for 0–TTC)

G1322	122	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
G1323	123								
G1324	124	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G1325	125	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G1326	126	CDZ	SMZ	*OV16	OVC			HIZ	HIX
G1327	127			DNCI			*ABSM	MIZ	IGNVRY
G1328	128							ITZ	ITX
G1329	129								
G1330	130	UI7	UI6	UI5	UI4	UI3	UI2	UI1	UI0
G1331	131	UI15	UI14	UI13	UI12	UI11	UI10	UI9	UI8
G1332	132	GOQSM	GOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0
G1333	133	WOSET						NOWT	
G1334	134			MFIN3	MFIN2	EXWT	EXSTP	EXRD	ENBKY
G1335	135								
G1336	136								
G1337	137								
G1338	138				–*EDCZ	*–EDCX		*+EDCZ	*+EDCX
G1339	139								
G1340	140					UINT			
G1341	141	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G1342	142								
G1343	143								
G1344	144								

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

G1345 145

--	--	--	--	--	--	--	--

G1346 146

--	--	--	--	--	--	--	--

G1347 147

--	--	--	--	--	--	--	--

Machine interface signal (CNC → PMC) (for tool post 2)

	#7	#6	#5	#4	#3	#2	#1	#0
F1348 148	OP	SA	STL	SPL			ZPZ	ZPX

F1349 149

MA		TAP	ENB	DEN		RST	AL
----	--	-----	-----	-----	--	-----	----

F1350 150

BF		DST		TF	SF		MF
----	--	-----	--	----	----	--	----

F1351 151

M28	M24	M22	M21	M18	M14	M12	M11
-----	-----	-----	-----	-----	-----	-----	-----

F1352 152

S28	S24	S22	S21	S18	S14	S12	S11
-----	-----	-----	-----	-----	-----	-----	-----

F1353 153

T28	T24	T22	T21	T18	T14	T12	T11
-----	-----	-----	-----	-----	-----	-----	-----

F1354 154

							SPAL
--	--	--	--	--	--	--	------

F1355 155

--	--	--	--	--	--	--	--

F1356 156

--	--	--	--	--	--	--	--

F1357 157

		MF3	MF2	M38	M34	M32	M31
--	--	-----	-----	-----	-----	-----	-----

F1358 158

AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
-----	-----	-----	-----	-----	-----	-----	-----

F1359 159

AR15	AR14	AR13	AR12	AR11	AR10	AR9	AR8
------	------	------	------	------	------	-----	-----

F1360 160

	WATO					ESEND	EREND
--	------	--	--	--	--	-------	-------

F1361 161

						ZP2Z	ZP2X
--	--	--	--	--	--	------	------

F1362 162

UO7	UO6	UO5	UO4	UO3	UO2	UO1	UO0
-----	-----	-----	-----	-----	-----	-----	-----

F1363 163

UO15	UO14	UO13	UO12	UO11	UO10	UO9	UO8
------	------	------	------	------	------	-----	-----

F1364 164

PRTSF	RWD						
-------	-----	--	--	--	--	--	--

F1365 165

PX7	PX6	PX5	PX4	PX3	PX2	PX1	PX0
-----	-----	-----	-----	-----	-----	-----	-----

F1366 166

--	--	--	--	--	--	--	--

Address list (for 0–TTC)

F1367	167	PZ7	PZ6	PZ5	PZ4	PZ3	PZ2	PZ1	PZ0
F1368	168								
F1369	169								
F1370	170								
F1371	171	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F1372	172	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F1373	173					R12O	R11O	R10O	R09O
F1374	174			HZO	HXO	ZRNO	MD4O	MD2O	MD1O
F1375	175	MP2O	MP1O	ROV2O	ROV1O	JV8O	JV4O	JV2O	JV1O
F1376	176	DRNO	MLKO	SBKO	BDTO	OV8O	OV4O	OV2O	OV1O
F1377	177					–ZO	+Z O	–XO	+X O
F1378	178	SPO	RTO	KEYO					
F1379	179								
F1380	180				BGEACT	RPALM	RPBSY	RPGDPL	INHKY
F1381	181								
F1382	182								
F1383	183								
F1384	184								
F1385	185								
F1386	186								
F1387	187								
F1388	188								
F1389	189								

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

F1390	190							
F1391	191							
F1392	192							
F1393	193	M228	M224	M222	M221	M218	M214	M212
F1394	194	M318	M314	M312	M311	M238	M234	M232
F1395	195	M338	M334	M332	M331	M328	M324	M322
F1396	196	UO107	UO106	UO105	UO104	UO103	UO102	UO101
F1397	197	UO115	UO114	UO113	UO112	UO111	UO110	UO109
F1398	198	UO123	UO122	UO121	UO120	UO119	UO118	UO117
F1399	199	UO131	UO130	UO129	UO128	UO127	UO126	UO125

Machine interface signal (PMC → CNC) (for tool post 2)

		#7	#6	#5	#4	#3	#2	#1	#0
G1400	200				PMC window DI				
G1401	201				PMC window DI				
G1402	202				PMC window DI				
									[Kind of data]
G1403	203				PMC window DI				
									[No. of words of data]
G1404	204				PMC window DI				
									[Axis number]
G1405	205				PMC window DI				
									[0 0]
G1406	206				PMC window DI				
G1407	207				PMC window DI				
G1408	208				PMC window DI				
G1409	209				PMC window DI				

Address list (for 0-TTC)

Machine interface signal (CNC → PMC) (for tool post 2)

		#7	#6	#5	#4	#3	#2	#1	#0
F1450	250				PMC window DO				
F1451	251				PMC window DO				
F1452	252				PMC window DO				
									[Transferred data]
F1453	253				PMC window DO				
									[Transferred data]
F1454	254				PMC window DO				
									[Transferred data]
F1455	255				PMC window DO				
									[Transferred data]
F1456	256				PMC window DO				
									[Transferred data]
F1457	257				PMC window DO				
									[Transferred data]
F1458	258				PMC window DO				
									[Transferred data]
F1459	259				PMC window DO				
									[Transferred data]
F1460	260				PMC window DO				
									[Transferred data]
F1461	261				PMC window DO				
									[Transferred data]
F1462	262				PMC window DO				
									[Transferred data]
F1463	263				PMC window DO				
									[Transferred data]
F1464	264				PMC window DO				
									[Transferred data]
F1465	265				PMC window DO				
									[Transferred data]

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

F1466	266				PMC window DO			
								[Transferred data]
F1467	267				PMC window DO			
								[Transferred data]
F1468	268				PMC window DO			
F1469	269				PMC window DO			
F1470	270							
F1471	271							
F1472	272							
F1473	273							
F1474	274							
F1475	275							
F1476	276	B7	B6	B5	B4	B3	B2	B1 B0
F1477	277	B15	B14	B13	B12	B11	B10	B9 B8
F1478	278	B23	B22	B21	B20	B19	B18	B17 B16
F1479	279	B31	B30	B29	B28	B27	B26	B25 B24

Address list (for CNC without PMC)

Address list (for CNC without PMC)

(1) M series

(a) MT → CNC (without PMC)

		#7	#6	#5	#4	#3	#2	#1	#0
X000	000								
		M18-36	M18-21	M18-5	M18-35	M18-20	M18-34	M18-19	M18-33
X002	002								
		M18-24	M18-8	M18-38	M18-23	M18-7	M18-37	M18-22	M18-6
X004	004	4NG							
		M18-11	M18-41	M18-26	M18-10	M18-40	M18-25	M18-9	M18-39
X006	006								
		M18-45	M18-14	M18-44	M18-13	M18-43	M18-12	M18-42	M18-27
X008	008	SKIP		*RILK			ZAE	YAE	XAE
		M18-49	M18-18	M18-48	M18-17	M18-47	M18-16	M18-46	M18-15
X010	010					/	/	/	/
		M20-11	M20-41	M20-26	M20-10				
X012	012								
		M20-45	M20-14	M20-44	M20-13	M20-43	M20-12	M20-42	M20-27
X014	014								
		M20-49	M20-18	M20-48	M20-17	M20-47	M20-16	M20-46	M20-15
X016	016	HX/ROV1	/	*DECX	/	-X	+X	SBK	BDT
		M1-6		M1-38		M1-20	M1-21	M1-11	M1-12
X017	017	HY/ROV2	/	*DECY	/	-Y	+Y	MLK	*ILK
		M1-7		M1-39		M1-22	M1-23	M1-9	M1-10
X018	018	HZ/DRN	/	*DECZ	/	-Z	+Z	/	/
		M1-8		M1-40		M1-24	M1-25		
X019	019	H4	/	*DEC4	/	-4	+4	/	/
		M20-40		M20-25		M20-9	M20-39		
X020	020	ZRN	*SSTP	SOR	SAR	FIN	ST	MP2	MP1/MINP
		M1-13	M1-37	M1-5	M1-14	M1-15	M1-16	M1-17	M1-18
X021	021	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
		M1-41	M1-26	M1-27	M1-19	M1-33	M1-34	M1-35	M1-36
X022	022	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
		M1-42	M1-43	M1-44	M1-45	M1-46	M1-47	M1-48	M1-49

7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

(b) CNC → MT (without PMC)

		#7	#6	#5	#4	#3	#2	#1	#0
Y048	048	OP	SA	STL	SPL	/	ZPZ/EF	ZPY	ZPX
		M2-5	M2-6	M2-7	M2-8		M2-27	M2-26	M2-25
Y049	049	MA	/	/	ENB	DEN	/	RST	AL
		M2-9			M2-41	M2-22		M2-23	M2-24
Y050	050	/	/	DST	/	TF	SF	/	MF
				M2-10		M2-20	M2-19		M2-21
Y051	051	M28	M24	M22	M21	M18	M14	M12	M11
		M2-33	M2-34	M2-35	M2-36	M2-37	M2-38	M2-39	M2-40
Y052	052	S28	S24	S22	S21	S18	S14/GR30	S12/GR20	S11/GR10
		M2-11	M2-12	M2-13	M2-14	M2-15	M2-16	M2-17	M2-18
Y053	053	T28	T24	T22	T21	T18	T14	T12	T11
		M2-42	M2-43	M2-44	M2-45	M2-46	M2-47	M2-48	M2-49
Y080	080								
		M19-8	M19-7	M19-6	M19-5	M19-4	M19-3	M19-2	M19-1
Y082	082								
		M19-16	M19-15	M19-14	M19-13	M19-12	M19-11	M19-10	M19-9
Y084	084					ZP4			
		M20-36	M20-21	M20-5	M20-35	M20-20	M20-34	M20-19	M20-33
Y086	086								
		M20-24	M20-8	M20-38	M20-23	M20-7	M20-37	M20-22	M20-6

(2) T series

(a) MT → CNC (without PMC)

		#7	#6	#5	#4	#3	#2	#1	#0
X000	000								
		M18-36	M18-21	M18-5	M18-35	M18-20	M18-34	M18-19	M18-33
X002	002								
		M18-24	M18-8	M18-38	M18-23	M18-7	M18-37	M18-22	M18-6
X004	004								
		M18-11	M18-41	M18-26	M18-10	M18-40	M18-25	M18-9	M18-39
X006	006								
		M18-45	M18-14	M18-44	M18-13	M18-43	M18-12	M18-42	M18-27

Address list (for CNC without PMC)

X008	008	SKIP						ZAE	XAE
		SKIP			SKIP4	SKIP3	SKIP2	ZAE	XAE
M18-49 M18-18 M18-48 M18-17 M18-47 M18-16 M18-46 M18-15									
Lower stage for 0-G									
X010	010					/	/	/	/
		M20-11	M20-41	M20-26	M20-10				
X012	012								
		M20-45	M20-14	M20-44	M20-13	M20-43	M20-12	M20-42	M20-27
X014	014								
		M20-49	M20-18	M20-48	M20-17	M20-47	M20-16	M20-46	M20-15
X016	016	HX/ROV1	/	*DECX	/	-X	+X	SBK	BDT
		M1-6		M1-38		M1-20	M1-21	M1-11	M1-12
X017	017	HY/ROV2	/	*DECY	/	-Z	+Z	MLK	MP1/MINP
		M1-7		M1-39		M1-22	M1-23	M1-9	M1-10
X018	018	DRN	/	*+LZ	/	GR2	GR1	/	/
		M1-8		M1-40		M1-24	M1-25		
X019	019	*DEC3	/	*DEC4	/			/	/
		M20-40		M20-25		M20-9	M20-39		
X020	020	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIX
		M1-13	M1-37	M1-5	M1-14	M1-15	M1-16	M1-17	M1-18
X021	021	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
		M1-41	M1-26	M1-27	M1-19	M1-33	M1-34	M1-35	M1-36
X022	022	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
		M1-42	M1-43	M1-44	M1-45	M1-46	M1-47	M1-48	M1-49

7


(b) CNC → MT (without PMC)

		#7	#6	#5	#4	#3	#2	#1	#0
Y048	048	OP	SA	STL	SPL	/		ZPT	ZPX
		M2-5	M2-6	M2-7	M2-8		M2-27	M2-26	M2-25
Y049	049	MA	/	/	ENB	DEN	/	RST	AL
		M2-9			M2-41	M2-22		M2-23	M2-24
Y050	050	/	/	DST	/	TF	SF	/	MF
				M2-10		M2-20	M2-19		M2-21

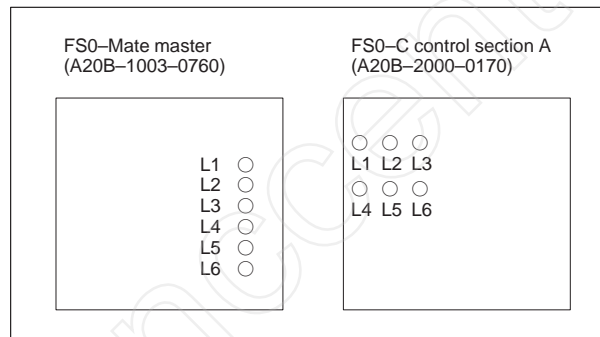
7. STATUS DISPLAY BY SELF DIAGNOSTIC DISPLAY

Y051	051	M28	M24	M22	M21	M18	M14	M12	M11
		M2-33	M2-34	M2-35	M2-36	M2-37	M2-38	M2-39	M2-40
Y052	052	S28	S24	S22	S21	S18	S14	S12	S11
		M2-11	M2-12	M2-13	M2-14	M2-15	M2-16	M2-17	M2-18
Y053	053	T28	T24	T22	T21	T18	T14	T12	T11
		M2-42	M2-43	M2-44	M2-45	M2-46	M2-47	M2-48	M2-49
Y080	080								
		M19-8	M19-7	M19-6	M19-5	M19-4	M19-3	M19-2	M19-1
Y082	082								
		M19-16	M19-15	M19-14	M19-13	M19-12	M19-11	M19-10	M19-9
Y084	084								
		M20-36	M20-21	M20-5	M20-35	M20-20	M20-34	M20-19	M20-33
Y086	086								
		M20-24	M20-8	M20-38	M20-23	M20-7	M20-37	M20-22	M20-6

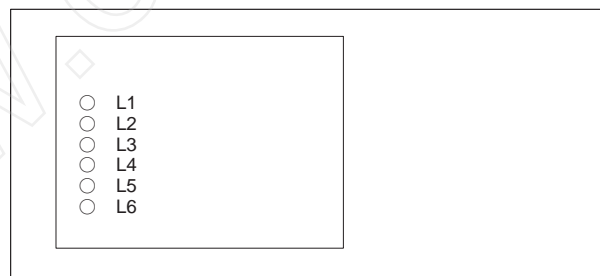
LEDs on the PC board

LED		Meaning and action to be taken
Number	Color	
L1	Green	Blinks during automatic operation. This is not an alarm.
L2	Red	Lights on any alarm. Take a necessary action according to the alarm number displayed on the CRT screen.
L3	Red	No memory card is installed.
L4	Red	(1) A watchdog timer alarm occurred. The master PC board or memory card is defective.  Equivalent to alarm 920 (2) A servo alarm occurred. (3) An axis card is not installed or is defective.
L5	Red	(1) A sub-CPU watchdog timer alarm occurred. Replace the sub-CPU PC board. (2) A servo alarm occurred on the fifth or sixth axis.
L6	Red	A system alarm occurred. (1) The analog interface card is defective. (2) The DNC1 card is defective. (3) The seventh- or eighth-axis card is defective.

(1) FS0-C control section A master (A20B-1002-0360)

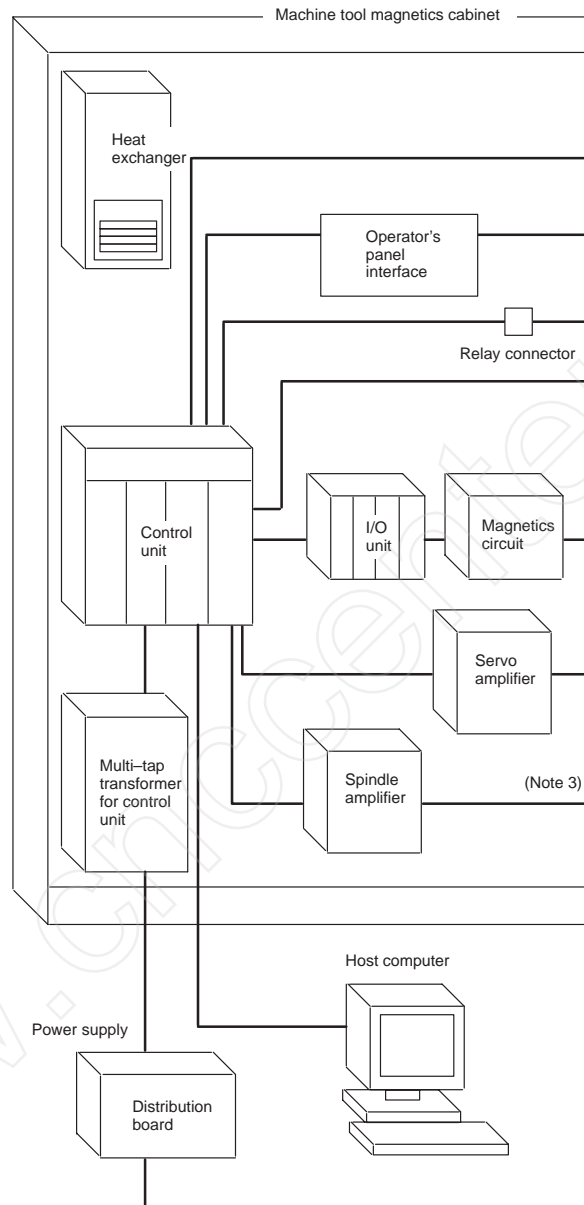


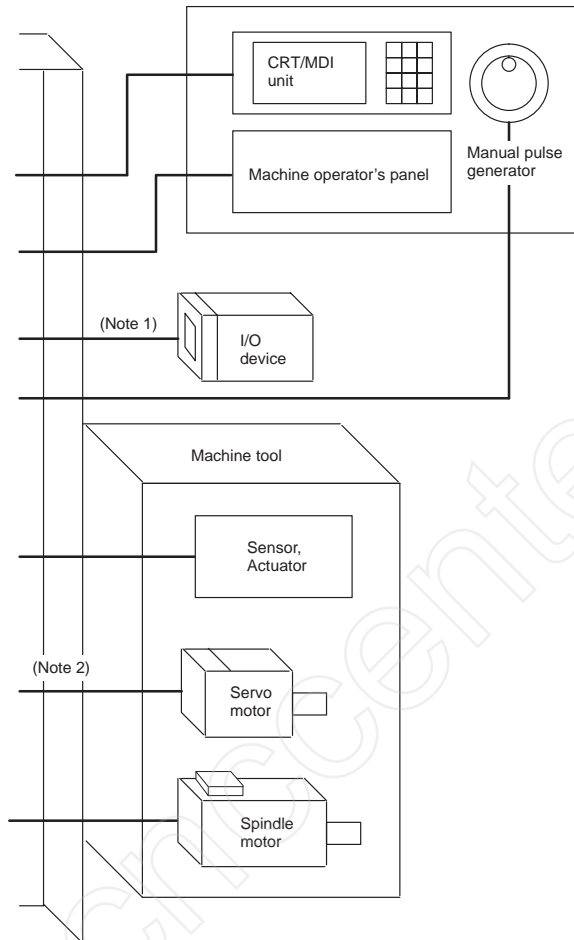
(2) FS0-C control section B master (A20B-1003-0750) (A20B-2000-0180)



8. HARDWARE

8.1 Configuration of CNC Machine Tool





8

NOTE 1 Refer to FANUC I/O Unit-MODEL A CONNECTION/MAINTENANCE MANUAL (B-61813E)

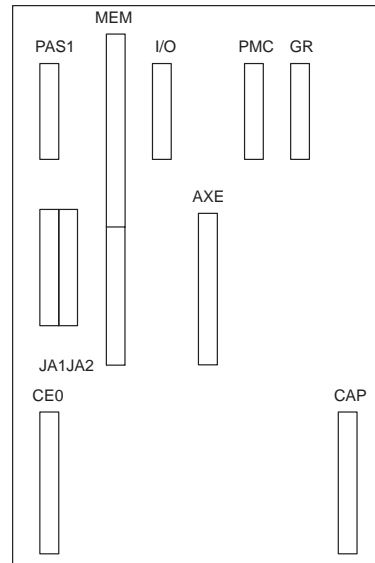
NOTE 2 Refer to FANUC AC SERVO MOTOR α series DESCRIPTIONS (B-65142E)

NOTE 3 Refer to FANUC AC SPINDLE MOTOR α series SERIAL INTERFACE DESCRIPTIONS (B-65152E)

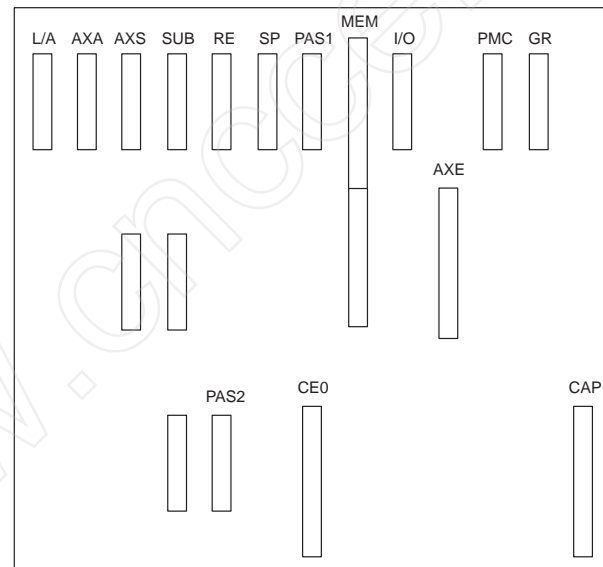
8. HARDWARE

8.2 Configuration of Control Unit

(1) Connector name of master PCB



Control unit A



Control unit B

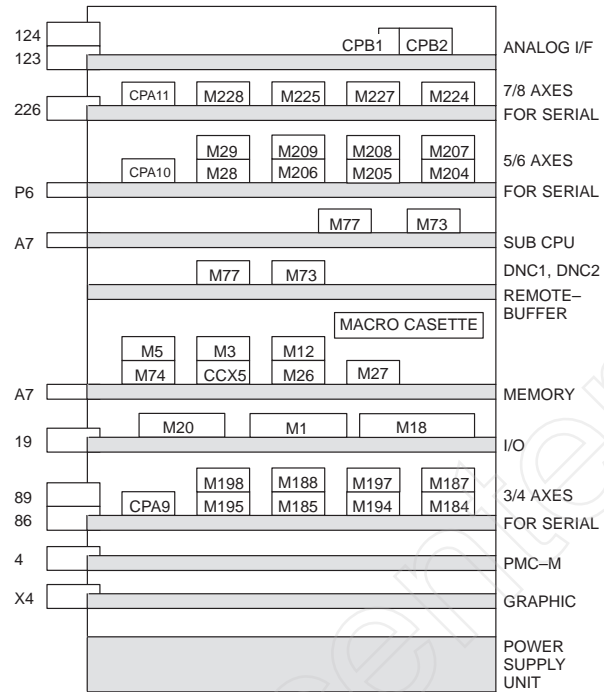
(2) Configuration of PCB

Connector name	PCB name			Specification
-	Master PCB	Control unit A	0-C	A20B-2000-0170
			0-Mate C	A20B-1003-0760 A20B-2000-0480
		Control unit B	0-C	A20B-2001-0060 A20B-2001-0110 A20B-2000-0180
			00-C	A20B-2000-0490
CAP	Power unit	A		A16B-1211-0850
		AI		A16B-1212-0100
		B2		A16B-1212-0110
		C		A16B-1211-0890
GR	Graphic PCB			A16B-1211-0920 A16B-2200-0350
	Manual pulse generator interface	For two or three manual pulse generators		A16B-1210-0801
PMC	PMC-M PCB	Electrical interface	ROM	A16B-1211-0901
			RAM	A16B-1211-0907
		Optical interface	ROM	A16B-1211-0903
			RAM	A16B-1211-0909
		I/O Link Master only	ROM	A16B-2200-0341
			RAM	A16B-2200-0346
AXE	Axis control PCB (2 to 3/4 axes)	0-Mate C For A/B phase pulse coder	2 axes	A16B-2200-0221
			3/4 axes	A16B-2200-0220
		0-C For A/B phase pulse coder	2 axes	A16B-2200-0361
			3/4 axes	A16B-2200-0360
		0-Mate C/0-C Response for serial pulse coder	2 axes	A16B-2200-0391
			3/4 axes	A16B-2200-0390
I/O	I/O PCB	I/O-C5 (40/40) Sink output type		A16B-1212-0222
		I/O-C6 (80/56) Sink output type		A16B-1212-0221
		I/O-C7 (104/72) Sink output type		A16B-1212-0220
		I/O-E1 (40/40) Source output type		A16B-1211-0972
		I/O-E2 (80/56) Source output type		A16B-1211-0971
		I/O-E3 (104/72) Source output type		A16B-1211-0970
MEM	Memory PCB	Response for serial spindle		A16B-1212-0215 A16B-1212-0216 A16B-2201-0101
		For analog spindle		A16B-1212-0210 A16B-2201-0103

8. HARDWARE

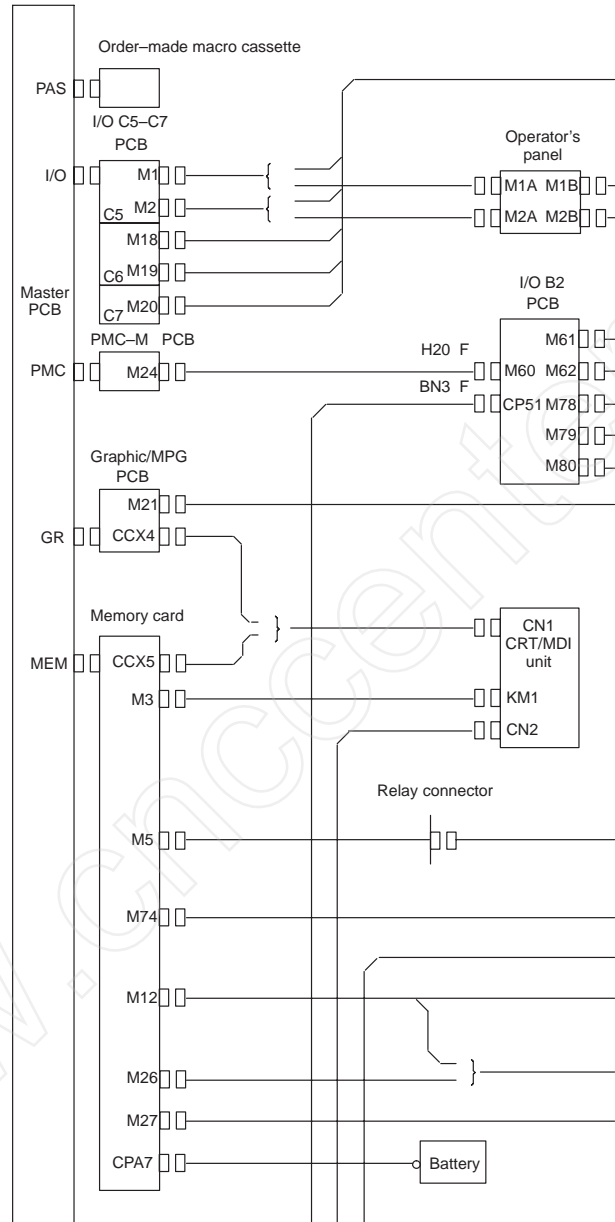
Connector name	PCB name		Specification
PAS1 (PAS2)	Cassette for macro executor	64KB	A02B-0091-C111
		128KB	A02B-0091-C113
		256KB	A02B-0091-C115
	Main side PAS1 (Sub side PAS2)	512KB	A02B-0098-C117
		1MB	A02B-0098-C118
JA1 JA2	DNC1 card (for control unit A)	For 0-Mate C	A16B-2200-0771
		For 0-C	A16B-2200-0776
	DNC2 card Remote buffer card (for control unit A)	For 0-C	A16B-2200-0775
SP	DNC1 card (for control unit B)	For 0-C	A16B-2200-0776
	DNC2 card Remote buffer card (for control unit B)	For 0-C	A16B-2200-0775
SUB	Sub PCU PCB	Standard (SUB CPU 16 bit)	A16B-2200-0320
		High-speed (SUB CPU 32 bit)	A16B-2201-0120
AXS	Axis control PCB (5/6 axes)	For A/B phase pulse coder	5/6 axes A16B-2200-0371
		Response for serial pulse coder	5/6 axes A16B-2200-0800
AXA	Axis control PCB (7/8 axes)	For A/B phase pulse coder	7 axes A16B-2200-0381
			7/8 axes A16B-2200-0380
		Response for serial pulse coder	7 axes A16B-2200-0791
			7/8 axes A16B-2200-0790
LA	Analog interface PCB		A16B-1211-0961
	Laser interface PCB		A16B-1212-0270

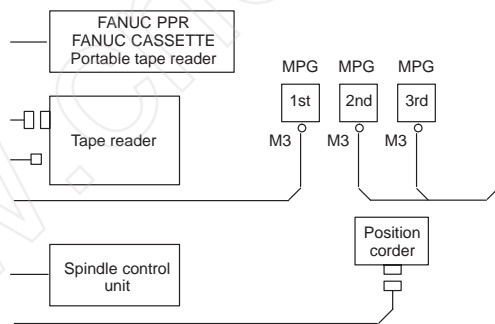
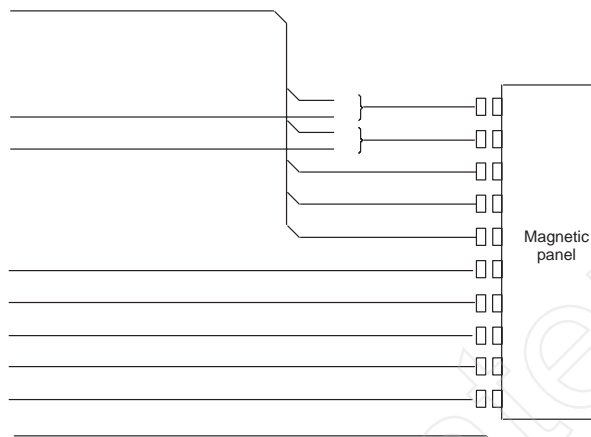
(3) Connector name



8. HARDWARE

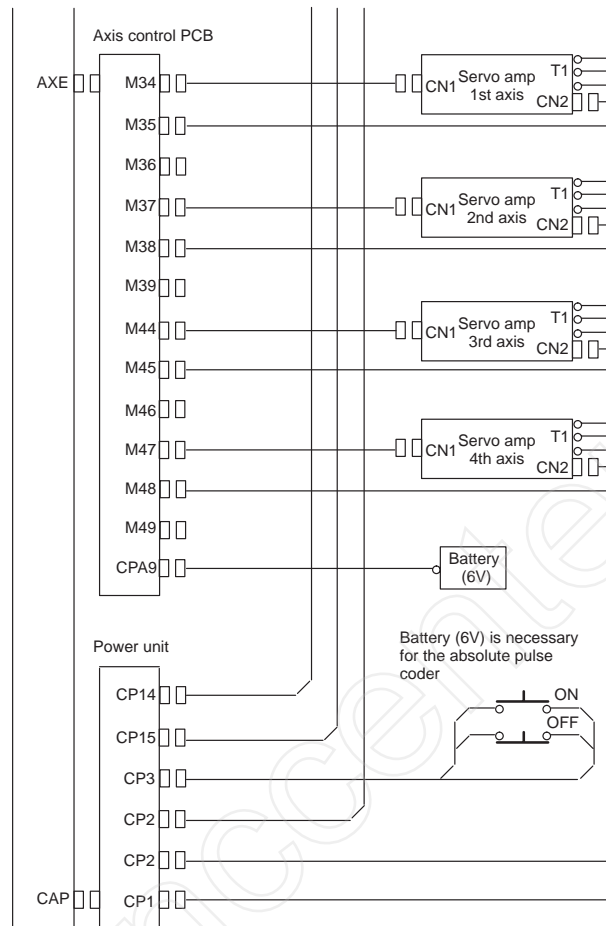
Overall machine-to-machine connection diagram (phase A/B pulse coder capability axis card with no separate detector)

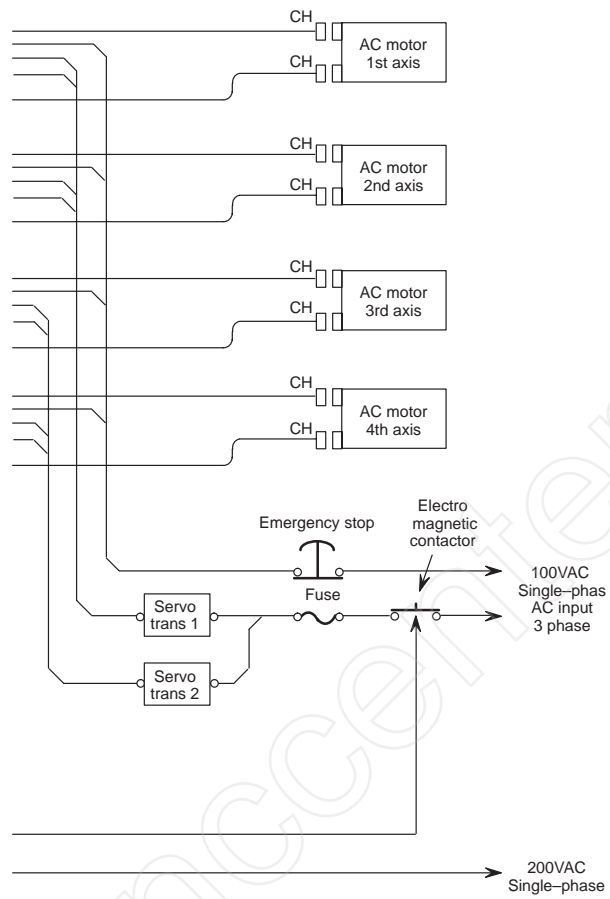




8

8. HARDWARE

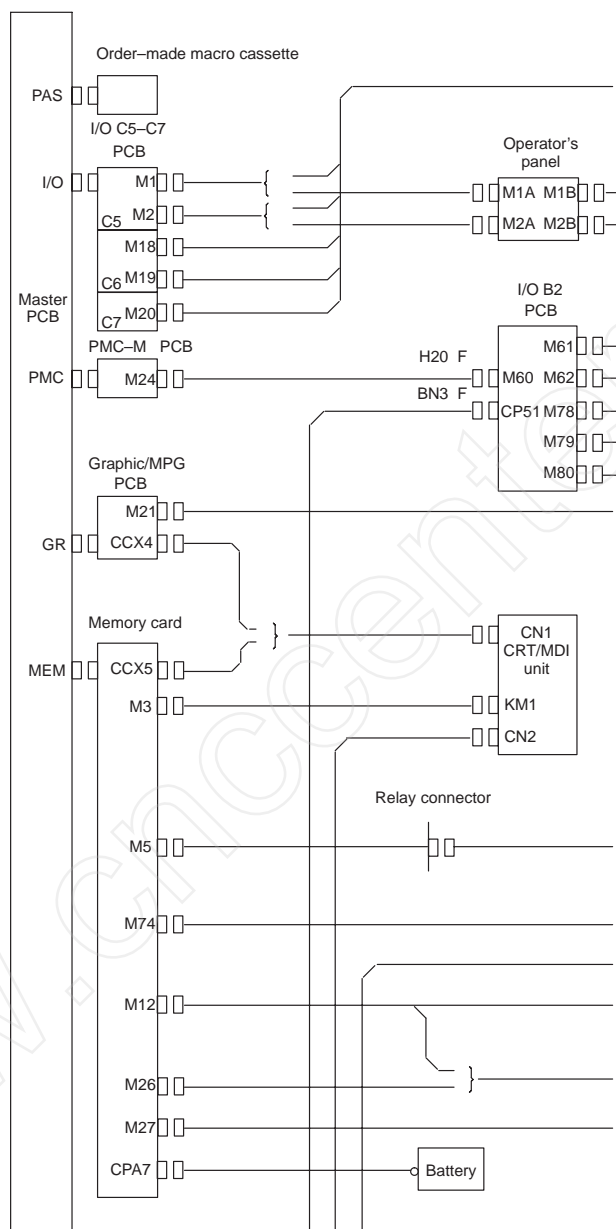


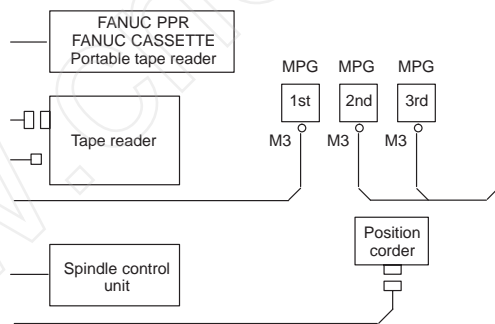
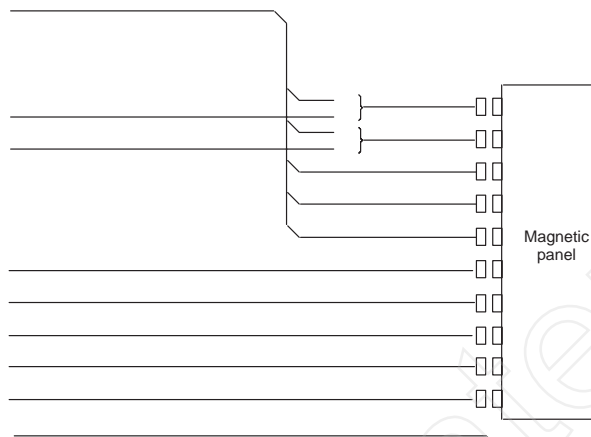


8

8. HARDWARE

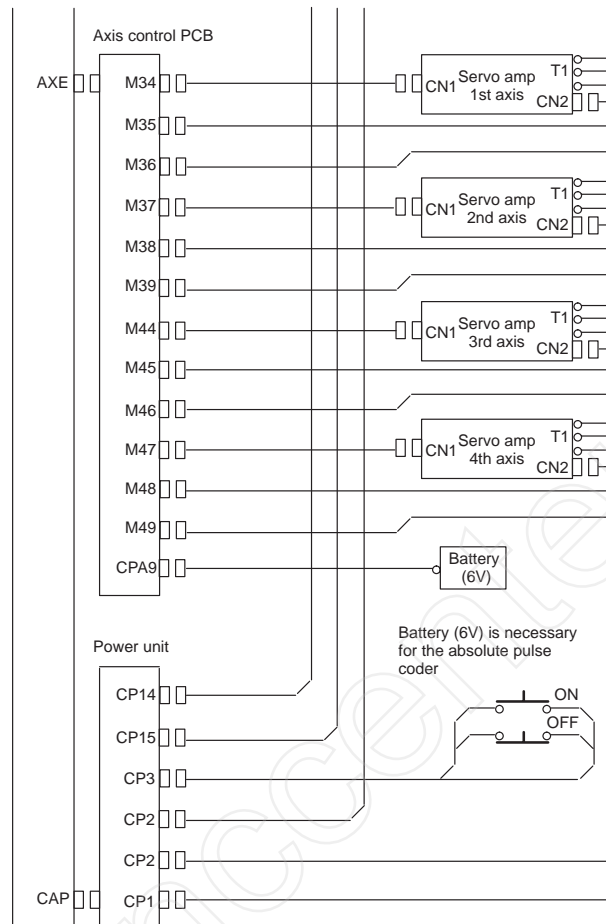
Overall machine-to-machine connection diagram (dedicated phase A/B pulse coder axis card with a separate detector)

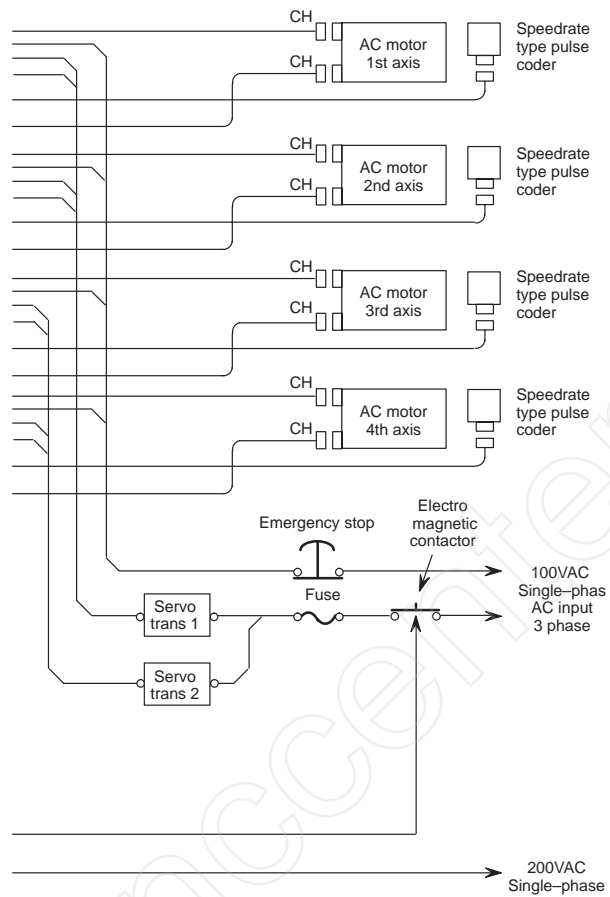




8

8. HARDWARE

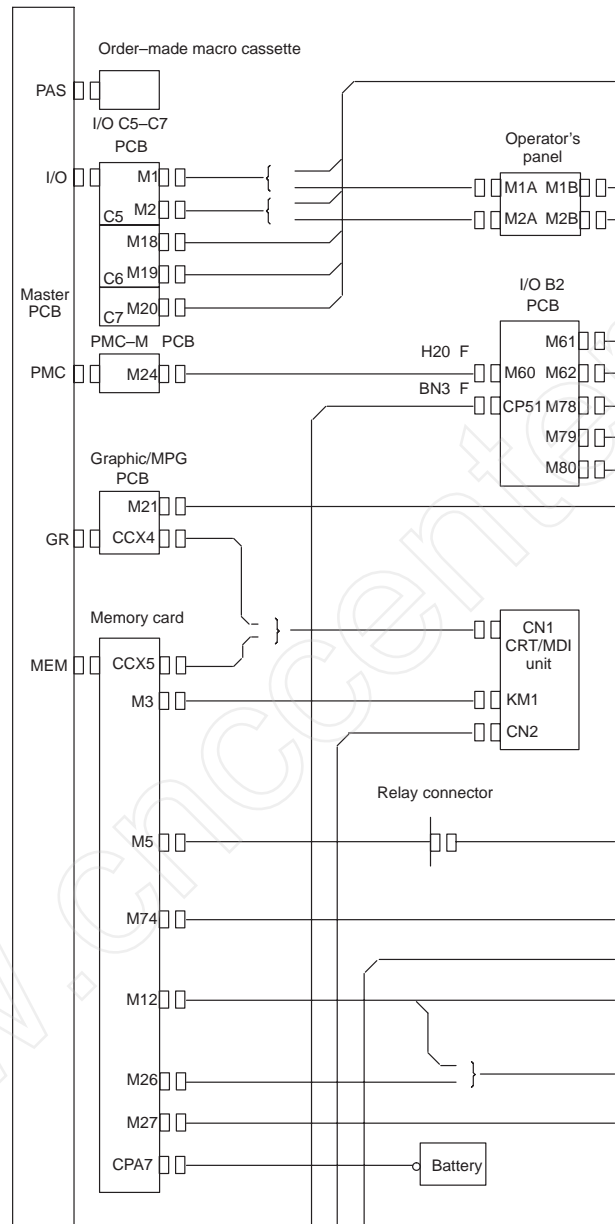


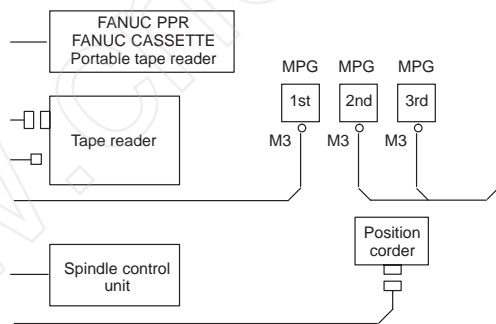
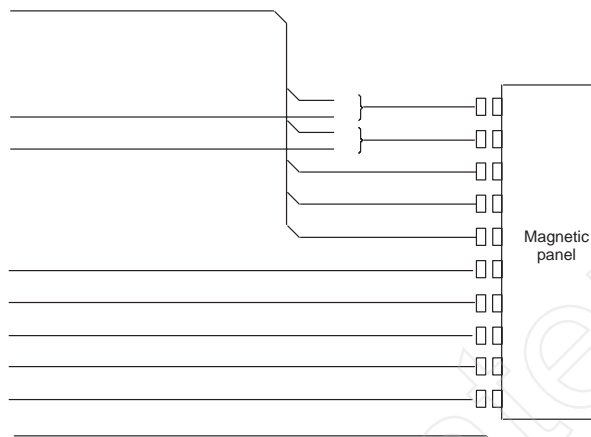


8

8. HARDWARE

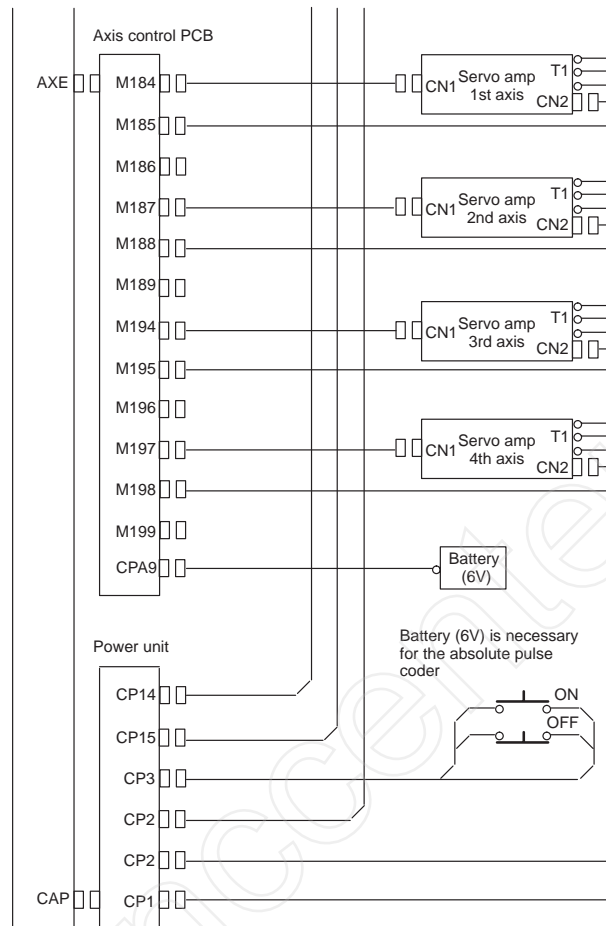
Overall machine-to-machine connection diagram (serial pulse coder capability axis card with no separate detector)

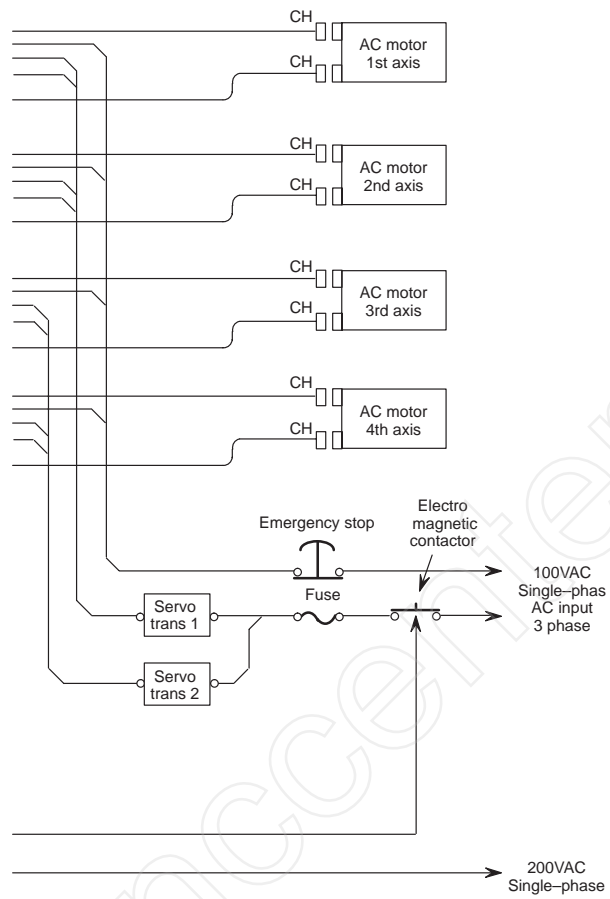




8

8. HARDWARE

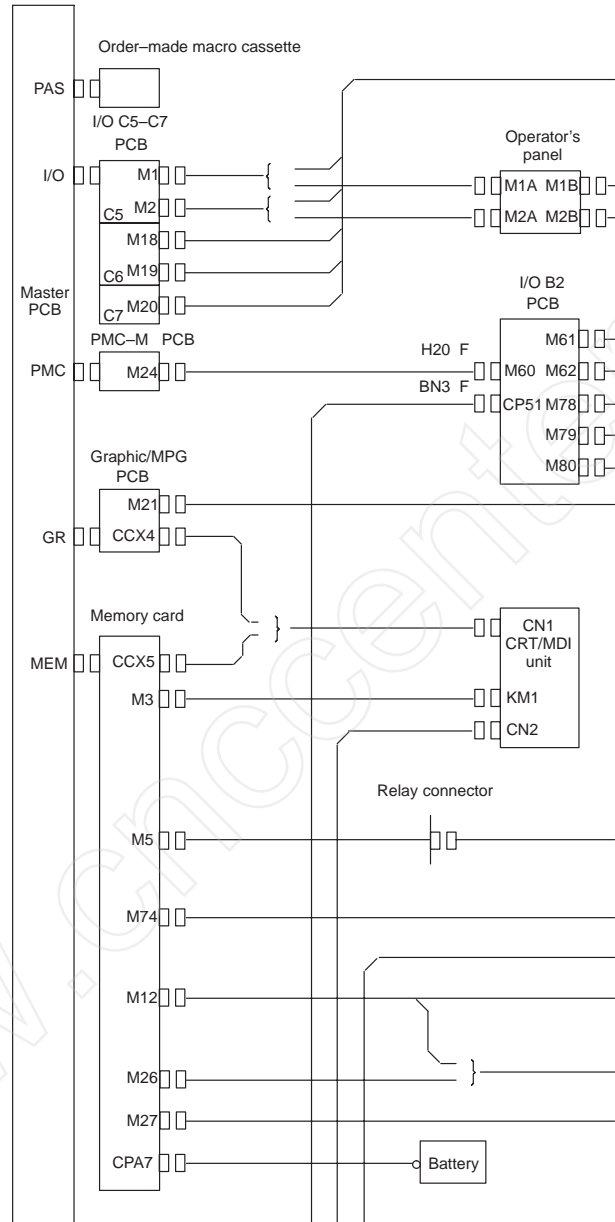


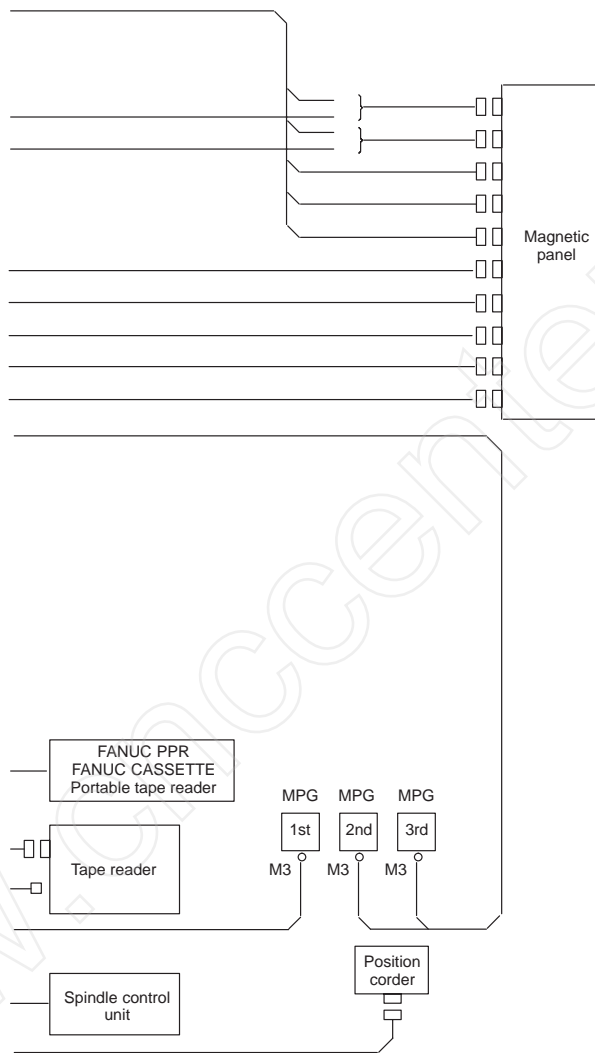


8

8. HARDWARE

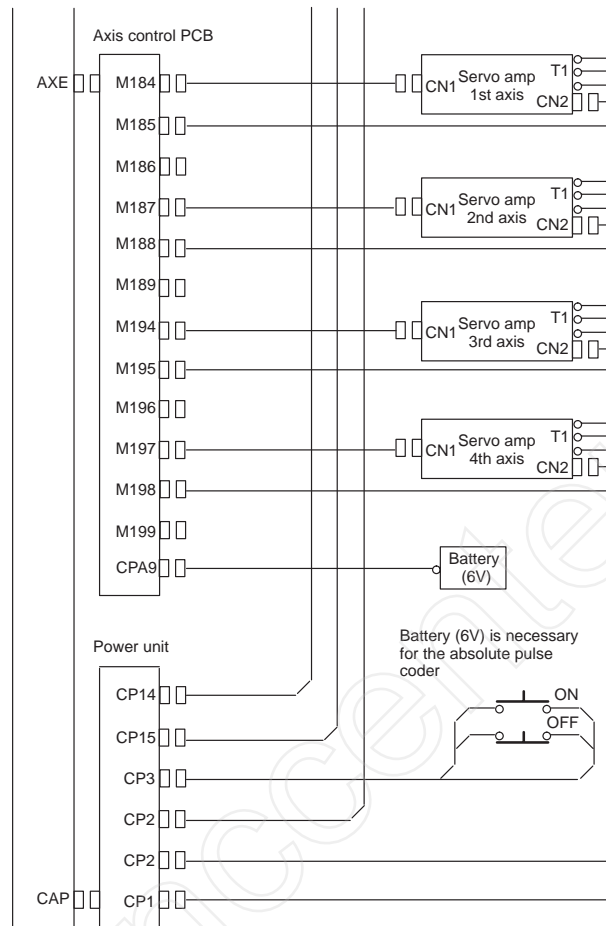
Overall machine-to-machine connection diagram (serial pulse coder capability axis card with a separate detector)

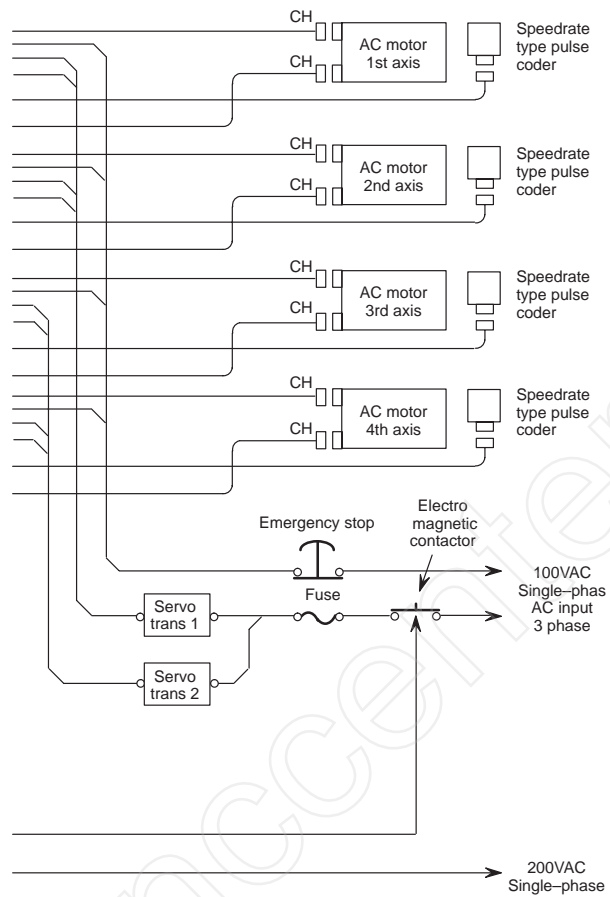




8

8. HARDWARE





FUNCTION PARAMETERS (FOR MACHINING CENTER)

- (1) Setting parameters
- (2) Parameters related to the reader/punch interface and remote buffer
- (3) Parameters related to controlled axes and the increment system
- (4) Parameters related to coordinate systems
- (5) Parameters related to the stroke limit
- (6) Parameters related to the feedrate
- (7) Parameters related to acceleration/deceleration control
- (8) Parameters related to servo motors
- (9) Parameters related to DI/DO
- (10) Parameters related to the CRT/MDI, display, and editing
- (11) Parameters related to programming
- (12) Parameters related to pitch error compensation
- (13) Parameters related to spindle control
- (14) Parameters related to tool compensation
- (15) Parameters related to canned cycles
- (16) Parameters related to rigid tapping
- (17) Parameters related to scaling and coordinate system rotation
- (18) Parameters related to unidirectional positioning
- (19) Parameters related to control in the normal direction
- (20) Parameters related to index table indexing
- (21) Parameters related to custom macro
- (22) Parameters related to automatic tool length measurement
- (23) Parameters related to graphic display
- (24) Parameters related to the display of operation time and number of parts
- (25) Parameters related to tool life management
- (26) Parameters related to the position switch function
- (27) Parameters related to manual handle feed/manual handle interrupt
- (28) Parameters related to the software operator's panel
- (29) Parameters related to program resumption
- (30) Parameters related to the high-speed cycle machining/high-speed remote buffer
- (31) Parameters related to PMC axis control
- (32) Parameters related to surface grinding machines (slanted axis control)
- (33) Parameters related to simple synchronous control
- (34) Parameters related to the PMC
- (35) Parameters related to the function for setting the zero point using the butt method
- (36) Parameters related to the DNC
- (37) Parameters related to the M-NET

NOTE 1 A bit parameter has two meanings, each described on the left and right of |. The meaning on the left applies when the bit is set to 0 and the meaning on the right applies when it is set to 1.

(Example) The current value is 0/1.

NOTE 2 For details, refer to "Description of Parameters".

(1) Setting parameters (1/1)

No.	Symbol	Description
0000	REVS, Y, 4	The mirror image of each axis is disabled/enabled.
0000	TVON	When a program is registered, a TV check is not executed/is executed.
0000	ISO	The code system used to output a program is EIA/ISO.
0000	INCH	The least input increment (input unit) of a program is one millimeter/one inch.
0000	I/O	Input/output unit on the reader/punch interface
0000	ABS	In the MDI mode, commands are specified with incremental values/absolute values.
0000	SEQ	Sequence numbers are not automatically inserted/are automatically inserted.
0000	PWE	Parameters cannot be written/can be written.
0000	TAPEF	The tape format is not converted/is converted.
0000	PRGNO	Number of the program to be stopped when the specified sequence number is found
0000	SEQNO	Number of the sequence to be stopped when it is found

(2) Parameters related to the reader/punch interface and remote buffer (1/2)

No.	Symbol	Description
0002#0 0012#0 0050#0 0051#0	STP2 STP2 STP2 STP2	(I/O=0) The stop bit is 1/2. (I/O=1) (I/O=2) (I/O=3)
0002#2 0012#0	ASR33 ASR33	(I/O=0) FANUC PPR etc./20-mA current interface (I/O=1)
0002#3 0012#3 0050#3 0051#3	RSASCI RSASCI RSASCI RSASCI	(I/O=0) The input/output code is ISO or EIA/ASCII. (I/O=1) (I/O=2) (I/O=3)
0002#7 0012#7 0050#7 0051#7	NFED NFED NFED NFED	(I/O=0) Beforehand and afterwards, a feed character is output/is not output. (I/O=1) (I/O=2) (I/O=3)
0015#5	PRWD	The rewind signal is not output/is output to the portable tape reader.
0018#6	TVC	In the comment section of a program, a TV check is executed/is not executed.
0038#1, #2 0038#4, #5 0038#6, #7	***, *** ***, *** ***, ***	(I/O=3) Setting the input/output unit (I/O=2) (I/O=0, 1)
0051#1	PARTY	(I/O=3) A parity bit is not provided/is provided.
0051#2	SYNAK	With protocol B, the output of the SYN and NAK codes is not controlled/is controlled.
0051#4	NCKCD	On the remote buffer interface, the CD (signal quality detection) status is monitored/is not monitored.

9. PARAMETERS

For M series

(2) Parameters related to the reader/punch interface and remote buffer (2/2)

No.	Symbol	Description
0051#5	ECKL	The baud rate clock of the remote buffer is an internal clock/external clock.
0055#0	ASCII	Communication of all data except NC data is executed in ISO code/ASCII code.
0055#1	ETX	The end code is CR/ETX.
0055#2	PROTCA	Communication protocol B/A
0055#3	RS42	The remote buffer interface is RS-232-C/RS-422.
0055#7	RMSTS	If the remote buffer interface is used with protocol A, the status of the remote buffer in the SAT message is always returned without specification (= 0)/host's request for switching the SET message is unconditionally returned.
0070#7	ICR	If data is output in ISO code, the EOB code is LF, CR, or CR/LF.
0075#7	IONUL	If a null character is detected while EIA codes are read, an alarm does not occur/occurs.
0076#5	PRWD	On reader/punch interface 2, rewinding is not executed/is executed.
0391#6	RS23BN	On the RS-232-C interface, the DC code is used/is not used.
0396#0	NCKER	The ER (RS -232C) or TR (RS-422) signal is checked/is not checked.
0396#2	BCC	The BCC value is checked/is not checked.
0396#3	ERCODE	A4-digit hexadecimal error code is not appended to a negative acknowledgement/is appended to a negative acknowledgement.
0399#6	FEDNUL	A feed character in a significant information section is output as a space character/null character.
0250	-----	(I/O=2) Baud rate
0251	-----	(I/O=3)
0552	-----	(I/O=0)
0553	-----	(I/O=1)

(3) Parameters related to controlled axes and the increment system (1/2)

No.	Symbol	Description
0001#0 7001#0	SCW SCWS	The least command increment (output unit) for a linear axis is one millimeter/inch.
0003#0 to #3 0066#4, #5 7003#0, #1	ZM*	(X to 4) The direction of a reference position return and initial direction are positive/negative. (7, 8) (5, 6)
0008#2 to #4 0280 0285, 0286 7130, 7131	ADW* ----- ----- -----	(4) Axis name (4) (7, 8) (5, 6)
0011#2 0062#4, #5 7032#0, #1	ADLN ADLIN* ROT*	(4) Rotation axis/linear axis (7, 8) Rotation axis/linear axis (5, 6) Linear axis/rotation axis
0024#1	LII10	The input/output unit is IS-B/IS-A.
0049#4	S3JOG	The number of axes that can be simultaneously controlled in manual operation is one/up to three.

(3) Parameters related to controlled axes and the increment system (2/2)

No.	Symbol	Description
0057#3	MIC	If a decimal point is omitted, the least input increment is not multiplied/is multiplied by ten.
0062#7	C8NG	The eighth axis is enabled/disabled.
0066#1	NBD78	If there are more servo axes than controlled axes, the single-axis extension function for the controlled axis is disabled/enabled.
0074#0 to #3	CRF*	If a reference position return is not made and a movement command other than G28 is specified, an alarm does not occur/occurs.
0076#1	JZRN	The function for setting the reference position without dogs is disabled/enabled.
0391#0 to #5	JZRN*	The function for setting the reference position without dogs is enabled/disabled.
0398#1	ROAX	The roll-over function is disabled/enabled.
0398#2	RODR	The direction of rotation specified in ABS mode is determined so that the distance of rotation is minimized/determined according to the sign of the specified value.
0398#3	ROCR	Relative coordinates which are not multiples of the travel distance per rotation are not rounded/are rounded.
0399#7	OUTZRN	If the remaining travel distance or active miscellaneous function is found at a manual return to the reference position, an alarm occurs/does not occur.
0269 to 0274	-----	Number of a servo axis
0279	-----	Attribute of the fourth axis
7032#7	ROT10S	If inch output is specified for the fifth or sixth axis, the unit of parameters related to feedrate is 0.1 deg/min/1 deg/min.

(4) Parameters related to coordinate systems (1/2)

No.	Symbol	Description
0010#7 7010#7	APRS APRSS	At a manual reference position return, automatic coordinate system setting is not executed/is executed.
0024#6	CLCL	At a manual reference position return, the local coordinate system is not canceled/is canceled.
0028#5	EX10D	When an external workpiece coordinate system shift is executed, the offset value is the input value/ten-times the input value.
0708 to 0711 0815 to 0818 0825, 0826 7708, 7709	----- ----- ----- -----	(X to 4) Coordinates of the reference position for automatic coordinate system setting (X to 4) (7, 8) (5, 6)
0735 to 0738 0866, 0867	----- -----	(X to 4) Distance from the first reference position to the second reference position (7, 8)
0751 to 0754	-----	Offset from the external workpiece reference point on each axis
0755 to 0758	-----	Offset from the first workpiece reference point on each axis (G54)

9. PARAMETERS

For M series

(4) Parameters related to coordinate systems (2/2)

No.	Symbol	Description
0759 to 0762	-----	Offset from the second workpiece reference point on each axis (G55)
0763 to 0766	-----	Offset from the third workpiece reference point on each axis (G56)
0767 to 0770	-----	Offset from the fourth workpiece reference point on each axis (G57)
0771 to 0774	-----	Offset from the fifth workpiece reference point on each axis (G58)
0775 to 0778	-----	Offset from the sixth workpiece reference point on each axis (G59)
0780 to 0783	-----	(X to 4) Distance from the first reference position to the third reference position
0868 to 0869	-----	(7, 8)
0784 to 0787	-----	(X to 4) Distance from the first reference position to the fourth reference position
0870, 0871	-----	(7, 8)
0860	-----	Travel distance per revolution about the rotation axis
7717	-----	Machine coordinate system when the fifth or sixth axis is a rotation axis

(5) Parameters related to the stroke limit (1/1)

No.	Symbol	Description
0008#6	OTZN	A Z-axis stored stroke check is executed/is not executed.
0015#4	LM2	The signal for switching the second stored stroke limit is disabled/enabled.
0024#4	INOUT	The inhibited area of the third stored stroke limit is the inside/outside.
0057#5	HOT3	The hardware OT signals +LMX to -LMZ (G129, #0 to #5) are disabled/enabled.
0065#3	PSOT	Before a reference position return is made, the stored stroke limit is checked/is not checked.
0076#7	OTRF0M	An alarm occurs after the stroke limit is exceeded/before the stroke limit is exceeded.
0700 to 0703	-----	(X to 4) First stored stroke limit in the positive direction on each axis
0821, 0822	-----	(7, 8)
7700, 7701	-----	(5, 6)
0704 to 0707	-----	(X to 4) First stored stroke limit in the negative direction on each axis
0823, 0824	-----	(7, 8)
7704, 7705	-----	(5, 6)
0743 to 0746	-----	Second stored stroke limit in the positive direction on each axis
0747 to 0750	-----	Second stored stroke limit in the negative direction on each axis
0804 to 0806	-----	Third stored stroke limit in the positive direction on each axis
0807 to 0809	-----	Third stored stroke limit in the negative direction on each axis

(6) Parameters related to the feedrate (1/2)

No.	Symbol	Description
0001#6 7001#6	RDRN RDRNS	For the rapid traverse command, a dry run is disabled/enabled.
0008#5	ROVE	The rapid traverse override signal ROV2 (G117, #7) is enabled/disabled.
0010#0	ISOT	When a reference position is not established, manual rapid traverse is disabled/enabled.
0011#3	ADNW	Feedrate specification A/B
0015#3	SKPF	With G31, a dry run, override, or automatic acceleration/deceleration is disabled/enabled.
0049#6	NPRV	When a position coder is not used, the command of feed per rotation is disabled/enabled.
0049#7	FML10	The unit of parameters in which the rapid traverse rate and cutting feedrate are specified is not multiplied/is multiplied by ten.
0059#0 to #2	EDP*	Of the commands specifying feed in the positive direction on each axis, external deceleration is valid for rapid traverse only/for both rapid traverse and cutting feed.
0059#3 to #5	EDM*	Of the commands specifying feed in the negative direction on each axis, external deceleration is valid for rapid traverse only/for both rapid traverse and cutting feed.
0393#1	COVOUT	The function for changing the speed of outer arc machining is disabled/enabled.
0393#4	HLCLMP	In helical interpolation, the feedrate is not clamped/is clamped to the maximum cutting feedrate.
0393#5	STOV0	If the cutting feedrate override is 0% in rapid traverse, a stop does not occur/occurs.
0397#2	OVR255	The signal of feedrate override in 1% steps is disabled/enabled.
0213	-----	Minimum deceleration ratio for the speed of inner arc machining by automatic corner override (MDR)
0214	-----	Amount of automatic override for the inside of a corner
0215	-----	Inside angle for determining whether to use automatic override for the inside of a corner
0216	-----	Feedrate change corresponding to a graduation on the manual pulse generator when the F command is specified with one digit
0482	-----	Feedrate after acceleration/deceleration for automatic corner deceleration
0483	-----	Difference between feedrates of two blocks, for each axis, for automatic corner deceleration
0487	-----	Feedrate after acceleration/deceleration for automatic corner deceleration (for look-ahead control)
0495	-----	Maximum feedrate with standard arc radius
0496	-----	Minimum feedrate with standard arc radius
0518 to 0521 0643, 0644 7518, 7519	----- ----- -----	(X to 4) Rapid traverse feedrate for each axis (7, 8) (5, 6)
0527	-----	Maximum cutting feedrate

9. PARAMETERS

For M series

(6) Parameters related to the feedrate (2/2)

No.	Symbol	Description
0530 7530	-----	(X to 4) FL speed in exponential acceleration/ deceleration of cutting feed (5, 6)
0533 7533	----- -----	(X to 4) F0 speed of rapid traverse override (5, 6)
0534 7534	----- -----	(X to 4) FL speed at a reference position re- turn (5, 6)
0548	-----	FL speed in exponential acceleration/decelera- tion of manual feed
0549	-----	Cutting feedrate in the automatic mode at power- on
0559 to 0562	-----	Jog rapid traverse rate for each axis
0565, 0566	-----	Jog feedrate when the rotary switch is set to posi- tion 10
0567	-----	Maximum cutting feedrate with feedrate specifi- cation B
0568	-----	F0 speed of rapid traverse override on an addi- tional axis with feedrate specification B
0569	-----	FL speed at a reference position return on an additional axis with feedrate specification B
0538, 0584	-----	Maximum feedrate by the F command specified with one digit
0580	-----	Start distance (Le) for inner corner automatic override
0581	-----	End distance (Ls) for inner corner automatic override
0605 to 0608	-----	FL speed of exponential acceleration/decelera- tion for manual feed on each axis
0636	-----	External deceleration speed
0684	-----	Feedrate with which the error detect function as- sumes that deceleration is completed
0788 to 0796	-----	Feedrate corresponding to the F command spe- cified with one digit
0878	-----	FL speed for exponential acceleration/decelera- tion of cutting feed (for look-ahead control)

(7) Parameters related to acceleration/deceleration control (1/2)

No.	Symbol	Description
0020#5	NCIPS	A position check is executed/is not executed.
0045#3	LSUP	For cutting feed, exponential acceleration/deceleration is executed/linear acceleration/deceleration after interpolation is executed.
0045#4	CCINP	The in-position width for cutting feed is specified with the same parameters as those used for rapid traverse (No. 0500 to 0503)/with different parameters (No. 0609 to 0612)
0048#4	SMZCT	Rapid traverse block overlap is disabled/enabled.
0076#0	ERDT	The error detect function is disable/enabled.
0379	-----	Feedrate ratio at which the next block is started for block overlap
0393#0	FERDT	Automatic corner deceleration is disabled/enabled.
0395#3	CHEAFD	For automatic corner deceleration, control based on the feedrate difference is disabled/enabled.
0395#6	FWBTYP	Linear acceleration/deceleration prior to interpolation is type A/type B.
0399#2	RPDFF	Feed forward control is applied only to cutting feed/applied to both cutting feed and rapid traverse.
0399#4	CINPS	For feed-type-based in-position check (CCINP: bit 4 of parameter 0045), the inposition width for cutting feed is specified using parameters other than those for rapid traverse, only when the next block also specifies cutting feed/regardless of the type of feed specified in the next block.
0480	-----	Parameter 1 for setting the acceleration for linear acceleration/deceleration prior to interpolation
0481	-----	Parameter 2 for setting the acceleration for linear acceleration/deceleration prior to interpolation
0482	-----	Feedrate after acceleration/deceleration for automatic corner deceleration
0483	-----	Difference between feedrates of two blocks, for each axis, for automatic corner deceleration
0485	-----	Parameter 1 for setting the acceleration for linear acceleration/deceleration prior to interpolation (for look-ahead control)
0486	-----	Parameter 2 for setting the acceleration for linear acceleration/deceleration prior to interpolation (for look-ahead control)
0522 to 0525	-----	(X to 4) Time constant of linear acceleration/deceleration in rapid traverse on each axis
0645, 0646	-----	(7, 8)
7522, 7523	-----	(5, 6)
0529	-----	Time constant of exponential acceleration/deceleration in cutting feed or manual feed
7529	-----	(5, 6)
0601 to 0604	-----	(X to 4) Time constant of exponential acceleration/deceleration in manual feed on each axis
0635	-----	Time constant of linear acceleration/deceleration after interpolation in cutting feed

9. PARAMETERS

For M series

(7) Parameters related to acceleration/deceleration control (2/2)

No.	Symbol	Description
0651 to 0656	-----	(X to 8) Time constant of exponential acceleration/deceleration in cutting feed on a PMC axis
7651, 7652	-----	(5, 6)
0863	-----	Standard arc radius for feedrate clamp according to arc radius
0864	-----	Critical angle between two blocks for automatic corner deceleration (for advanced preview control)
0865	-----	Critical angle between two blocks for automatic corner deceleration
0877	-----	Time constant for exponential acceleration/deceleration of cutting feed (for advanced preview control)
0879	-----	Time constant for linear acceleration /deceleration after interpolation of cutting feed (for advanced preview control)
0952 to 0955	-----	(X to 4th) Time constant T2 for rapid traverse bell-shaped acceleration/deceleration for each axis

(8) Parameters related to servo motors (1/2)

No.	Symbol	Description
0004 to 0007	GRD*	(X to 4) Capacity of the reference counter of each axis
0067, 0068	GRD*	(7, 8)
0570 to 0575	-----	(X to 8)
7004, 7005	DMR*	(5, 6)
7570, 7571	-----	(5, 6)
0004 to 0007	DMR*	(X to 4) Detection multiplication of each axis (DMR)
0067, 0068	DMR*	(7, 8)
7004, 7005	DMR*	(5, 6)
0010#2	OFFVY	If VRDY is set to 1 before PRDY is set to 1, an alarm occurs/does not occur.
7010#2	OFFVYS	
0021#0 to #5	APC*	(X to 8) For each axis, an absolute-position detector is not used/is used.
7021#0, #1	APC*	(5, 6)
0021#6	NOFLUP	At power-on before the reference position is established by the absolute-position detector, a follow-up is executed/is not executed.
0022#0 to #5	ABS*	(X to 8) The reference position has not yet been established/has already been established by the absolute-position detector.
7022#0, #1	ABS*	(5, 6)
0035#7	ACMR	(X to 8) An optional CMR is not used/is used.
7035#7	ACMRS	
0037#0 to #5	SPTP*	(X to 8) As a position detector, a separate pulse coder is not used/is used.
7037#0, #1	SPTP*	(5, 6)
0037#7	PLCO1	(X to 8) A high-resolution pulse coder is not used/is used.
7037#7	PLCO1S	
0076#4	ADBLS	Cutting feed and rapid traverse separate backlash is disabled/enabled.

(8) Parameters related to servo motors (2/2)

No.	Symbol	Description
0390#0 to #5	NREQ*	If an absolute pulse coder is used for each axis and the zero point is not established before power-on, an alarm requesting a return to the reference position does not occur/occurs.
0399#3	SFDEC	The reference position shift function is disabled/enabled.
0399#5	FUNO	If a servo alarm is detected by the CNC, a follow-up is executed/is not executed.
0100 to 0103 0275, 0276 7100, 7101	----- ----- -----	(X to 4) Command multiplication of each axis (CMR) (7, 8) (5, 6)
0265	-----	Time interval for leveling the current value display on the servo adjustment screen
0452 to 0457 0739 to 0742 0827, 0828 7580, 7581 7739, 7740	----- ----- ----- ----- -----	(X to 8 3RD WORD) Count at the zero point of the APC (X to 4 LOWER 2 WORD) (7, 8 LOWER 2 WORD) (5, 6 3RD WORD) (5, 6 LOWER 2 WORD)
0500 to 0503 0637, 0638 7500, 7501	----- ----- -----	(X to 4) Effective area on each axis (7, 8) (5, 6)
0504, 7507 0639, 0640 7504, 7505	----- ----- -----	(X to 4) Limit on position error for each axis during traveling (7, 8) (5, 6)
0508 to 0511 0641, 0642 7508, 7509	----- ----- -----	(X to 4) Grid shift amount for each axis (7, 8) (5, 6)
0512 to 0515 0632, 0633	----- -----	(X to 4) Loop gain of position control for each axis (7, 8)
0517 7517	----- -----	(X to 4) Loop gain of position control common to all axes (5, 6)
0535 to 0538 0647, 0648 7535, 7536	----- ----- -----	(X to 4) Backlash compensation for each axis (7, 8) (5, 6)
0593 to 0596 0649, 0650 7593, 7594	----- ----- -----	(X to 4) Limit on position error for each axis during a stop (7, 8) (5, 6)
0609 to 0612	-----	(X to 4th) In-position width for cutting feed for each axis
0956	-----	Distance between the position at which the deceleration dog is turned off and the first grid point

9. PARAMETERS

For M series

(9) Parameters related to DI/DO (1/1)

No.	Symbol	Description
0001#2	DCS	The START key on the MDI panel is connected through the machine/is not connected through the machine.
0001#5	DECI	(X to 4) At a reference position return, deceleration occurs when the deceleration signal is set to 0/1.
7001#5	DECIS	(5, 6)
0003#4	OVRI	(X to 4) Acceleration occurs when the override signal or rapid traverse override signal is set to 0/1.
7003#4	OVRIS	(5, 6)
0008#7	EILK	Interlock occurs on all axes or Z-axis only/on an individual axis.
0009#0 to #3	TFIN	(X to 4) Time period in which the signal (FIN, bit 3 of G120) indicating that the miscellaneous function, spindle-speed function, or tool function has been completed is accepted
7009#0 to #3	TFIN	(5, 6)
0009#4 to #7	TMF	(X to 4) Period up to the time when the signal indicating that the code of the miscellaneous function, spindle-speed function, or tool function is read is sent
7009#4 to #7	TMF	(5, 6)
0012#1	ZILK	Interlock occurs on all axes/Z-axis only.
0015#2	RIKL	The high-speed interlock signal *RILK (X008, #5) is disabled/enabled.
0019#1	C4NG	The signal to ignore the fourth axis 4NG (X004, #7) is disabled/enabled.
0020#4	BCD3	For the B code, six digits/three digits are output.
0045#2	RWDOUT	The rewind signal RWD (F164, #6) is output only when the tape reader is rewinding/is output while a program in memory is rewound.
0045#7	HSIF	For processing the M, S, T, and B codes, the standard interface is used/the high-speed interface is used.
0049#0	DILK	The signal for separate interlock in the direction of each axis is disabled/enabled.
0049#1	RDIK	The high-speed interlock signal is always enabled/is disabled when the signal for separate interlock in the direction of each axis is set to 1.
0070#4	DSTBGE	When output is started in background editing, the signal for starting manual data input DST (F150, #5) is output/is not output.
0252	-----	Extension time for the reset signal

(10) Parameters related to the CRT/MDI, display, and editing (1/3)

No.	Symbol	Description
0001#1	PROD	In the relative coordinate display, tool length compensation is included/is not included.
0001#4	IOF	An offset value is input from the MID panel in the ABS mode/INC mode.
0002#1 7002#1	PPD PPDS	Coordinate system setting does not cause relative coordinates to be pre-set/causes relative coordinates to be pre-set.
0010#1	EBCL	When a program in memory is displayed, the EOB code is displayed as ;/.
0011#7	MCINP	By MINP (G120, #0), a program is not registered/is registered in memory.
0013#4	INHMN	Menus are displayed/are not displayed.
0015#0	CBLNK	The cursor blinks/does not blink.
0015#1	NWCH	The amount of tool wear compensation is displayed with a character W/without a character W.
0015#6	REP	If a program registered from the reader/punch interface has the same number as another program registered in memory, an alarm occurs/the program is replaced.
0018#5	PROAD	In the absolute coordinate display, tool length compensation is included/is not included.
0018#7	EDITB	With the standard keyboard, editing A is executed/editing B is executed.
0019#6	NEOP	M02, M30, or M99 terminates program registration/does not terminate program registration.
0019#7	DBCD	On the diagnostic screen, the data of a timer counter is displayed in binary/decimal.
0023#0 to #6	***	Setting of the language to be used on the display
0028#0	PRCPOS	On the program check screen, relative coordinates are displayed/absolute coordinates are displayed.
0028#2	DACTF	The actual speed is not displayed/is displayed.
0028#3	MMDL	In MDI operation B, the modal status is not displayed/is displayed.
0029#6	DSPSUB	On the absolute or relative position display screen, the fifth or sixth axis is not displayed/is displayed.
0035#0 to #3	NDSP*	The current position on each axis is displayed/is not displayed.
0038#3	FLKY	With the CRT or MDI, a standard keypad is used/full keypad is used.
0040#0	NAMPR	On the program directory screen, program names are not displayed/are displayed.
0040#4	SORT	On the program directory screen, programs are arranged in the order in which they are registered/in the ascending order of program numbers.
0045#0	RDL	Under I/O unit external control, reading depends on the REP bit (bit 6 of parameter 0015)/a program is registered after all programs are deleted.
0045#1	RAL	Reading on the reader/punch interface causes all programs to be registered/only the first program to be registered.

9. PARAMETERS

For M series

(10) Parameters related to the CRT/MDI, display, and editing (2/3)

No.	Symbol	Description
0048#7	SFFDSP	The soft-key display and control depend on the configuration of additional functions/are executed regardless of additional functions.
0050#1	NOFMK	When a sequence number is searched for, the format is checked/is not checked.
0056#0	NOCND	When the length of part program storage is 120 or 320 m, or if the background editing function is provided, memory is automatically compressed/ is not automatically compressed.
0057#0	PWNNC	At power-on, the MMC screen is displayed/CNC screen is displayed.
0057#1	ALMNC	If an alarm occurs in the CNC while the MMC screen is displayed, the MMC screen is displayed/CNC screen is displayed.
0057#2	SALNC	If a system alarm occurs in the CNC while the MMC screen is displayed, the MMC screen is displayed/CNC screen is displayed.
0060#0	DADRD	On the diagnostic screen, addresses are not displayed/are displayed.
0060#2	LDDSPG	Dynamic ladder display is not executed/is executed.
0060#3	C14DGN	When a 14-inch CRT is used, the diagnostic screen of the 9-inch format is displayed/14-inch format is displayed.
0060#5	OPMNDP	Operating monitor display is invalid/valid.
0060#6	EXTSP	The function to search for or display a protected program is disabled/enabled.
0060#7	IGNCOL	On a 9-inch high-resolution CRT, monochrome screens are displayed/color screens are displayed.
0061#6	DSP78	In absolute and relative coordinates on the current position display, the coordinate on the seventh or eighth axis is not included/is included.
0063#0	MTDSPI	Machine coordinates are not displayed according to the input system/are displayed according to the input system.
0063#1	PRSTIN	Automatic coordinate system setting in the inch input mode is specified in parameter 0708 and subsequent parameters/in parameter 0815 and subsequent parameters.
0064#0	SETREL	Pre-setting relative coordinates causes clearing to zero/optional values to be used.
0064#1	ALLPRE	For pre-setting relative coordinates, the standard specification is selected/axes are selected by numeric keys.
0064#3	IOMDGN	On the diagnostic screen, DI/DO of the I/O Link and an extended R/D area are not displayed/are displayed.
0064#5	NPA	If an alarm occurs or an operator message is input, the alarm or message screen is displayed instead/is not displayed.
0066#0	ALL56	On the complete current position screen, the fifth or six axis is not displayed/is displayed.
0076#2	IOP	Input or output of an NC program can be stopped by an NC reset/only by pressing the [STOP] soft key.

(10) Parameters related to the CRT/MDI, display, and editing (3/3)

No.	Symbol	Description
0077#2	SGD	Servo waveform display is invalid/valid.
0077#6	HLKEY	The MDI keys are not processed in the high-speed mode/are processed in the high-speed mode.
0078#0	NOINOW	The amount of tool wear compensation can be input with the MDI keys/cannot be input with the MDI keys.
0078#1	NOINOG	The amount of tool geometry compensation can be input with the MDI keys/cannot be input with the MDI keys.
0078#2	NOINMV	A macro variable can be input with the MDI keys/cannot be input with the MDI keys.
0078#3	NOINWZ	An offset from the workpiece reference point can be input with the MDI keys/cannot be input with the MDI keys.
0389#0	SRVSET	The servo setting screen is displayed/is not displayed.
0393#2	WKNMDI	If the automatic operation is started or halted, an offset from the workpiece reference point can be input with the MDI keys/cannot be input with the MDI keys.
0393#7	DGNWEB	If PWE is set to 0, a PMC parameter cannot be input/can be input.
0395#1	TLSCUR	On the offset screen, the cursor position is not retained/is retained.
0397#7	SERNAI	The contents of alarm 409 are not displayed/are displayed.
0398#0	WKINC	With the MDI keys, an offset from the workpiece reference point is input in the ABS mode/INC mode.
0337 to 0346	-----	Character codes of the title
0351 to 0355	-----	Character codes of the NC name
0550	-----	Increment used when the sequence number is automatically inserted
0797	-----	Encryption
0798	-----	Key

(11) Parameters related to programming (1/2)

No.	Symbol	Description
0010#4	PRG9	Editing of subprograms from O9000 to O9999 is not inhibited/is inhibited.
0011#6	G01	The mode selected at power-on is G00/G01.
0015#7	CPRD	If a decimal point is omitted, the selected unit is the least input increment/mm, inch, deg, and sec.
0016#3	NPRD	Decimal point input or display is used/is not used.
0019#5	M02NR	After M02 is executed, a return to the beginning of the program is made/is not made.
0028#4	EXTS	External program number search is invalid/valid.
0029#5	MABS	In MDI operation B, the ABS or INC command depends on setting ABS/G90 or G91.
0030#7	G91	The mode selected at power-on is G90/G91.

9. PARAMETERS

For M series

(11) Parameters related to programming (2/2)

No.	Symbol	Description
0045#6	CLER	Pressing the RESET key, setting the external reset signal or an emergency stop sets the system in the reset state/clear state.
0057#7	MBCLR	When a reset occurs, MDI-B programs are not cleared/are cleared.
0063#2	M198P	Address P of M198 is used with a file number/program number.
0065#7	M3B	A single block can contain only one M code/up to three M codes.
0389#2	PRG8	Editing of programs from O8000 to 8999 is not inhibited/is inhibited.
0391#7	NOCLR	In the clear state, a specific G code is cleared/is not cleared.
0393#3	M3PQNG	An M code specified with three digits is valid/invalid.
0393#6	RADCHK	When circular interpolation is specified, the difference between the radius values at the start point and end point is not checked/is checked.
0394#6	WKZRST	In workpiece coordinate system setting, an NC reset does not cause a return to G54/causes a return to G54.
0394#7	CAKEY	On the parameter, diagnostic, or offset screen, pressing the CAN key does not erase a single character/erases a single character.
0395#0	DLG99	In the feed per rotation mode, the dwell command is specified with a time period/spindle speed.
0396#7	EORRE	If EOR is read without the program end command, an alarm occurs/a reset occurs.
0111, 0112	-----	M code which is not buffered
0212	-----	Plane selected at power-on
0484	-----	Program number for parameter, macro variable, or diagnostic data
0634	-----	Program number registered in a simultaneous input/output operation
0876	-----	Limit of arc radius error

(12) Parameters related to pitch error compensation (1/1)

No.	Symbol	Description
0011#0, #1	PML*	(X to 4) Magnification of pitch error compensation
7011#0, #1	PML.S	(5, 6)
0712 to 0715	-----	(X to 4) Interval of pitch error compensation for each axis
7713, 7714	-----	(5, 6)
1000 to 6000	-----	Reference position of pitch error compensation for each axis
1001 to 6128	-----	Pitch error compensation for each axis

(13) Parameters related to spindle control (1/2)

No.	Symbol	Description
0003#5	GST	By SOR (G120, #5), spindle orientation is executed/a gear-change is executed.
0012#6	G84S	If the G74 or G84 cycle is specified, the gear is changed at a point specified through S analog output gear-change method A or B/at a point specified in parameters 0540 and 0556.
0013#5	ORCW	In spindle orientation, S analog output is positive/negative.
0013#6, #7	TCW, CWM	Sign output in S analog output
0014#0	SCTA	The spindle speed arrival signal is checked conditionally/always.
0020#7	SFOUT	SF (F150, #2) is output when a gear-change is made/even if a gear-change is not made.
0024#2	SCTO	The spindle speed arrival signal SAR (G120, #4) is not checked/is checked.
0028#6, #7	PSG※	Gear ratio between the spindle and position coder
0029#4	FSOB	Under constant surface speed control, SF (F150, #2) is output/is not output.
0035#6	LGCM	The gear-change speed is the maximum speed of each gear (method A)/is determined by parameters 0585 and 0586 (method B).
0041#4, #5	SSCA※	Axis used as the calculation standard under constant surface speed control
0062#3	SPMRPM	Parameters of spindle speed control are specified in units of 1 RPM/10 RPM.
0065#1	CZRN	For the first G00 command after the Cs contour control mode is selected, a reference position return is made, then positioning is executed/normal positioning is executed.
0070#0, #1	DAC※	For D/A conversion, the analog interface board is not used/is used.
0071#0	ISRLPC	When the serial interface spindle is used, the position coder signal is fetched from the optical fiber cable/connector M27.
0071#4	SRL2SP	The number of serial interface spindles connected is one/two.
0071#7	FSRSP	The serial interface spindle is not used/is used.
0080#2, #3	MORCM※	For the first and second spindle motors, the spindle orientation function with the stop position set externally is not used/is used.
0080#6, #7	SP※NEG	The directions of rotation of the first and second spindles during synchronous control of the spindle are the same as the specified sign/are the opposite of the specified sign.
0108	-----	Spindle speed in stable spindle rotation
0108	-----	Spindle motor speed at a gear-change
0110	-----	Delay timer if the spindle speed arrival signal SAR (G120, #4) is checked
0303	-----	Permissible difference in the number of error pulses for phasing in the synchronous control mode
0516	-----	Data for adjusting the gain under constant surface speed control (analog output)

9. PARAMETERS

For M series

(13) Parameters related to spindle control (2/2)

No.	Symbol	Description
0539 0541 0555	----- ----- -----	[Neutral gear] Maximum spindle speed [Low gear] [High gear]
0540 to 0543	-----	Spindle speed when the voltage for specifying the spindle speed of each gear is 10 V
0540 0556	----- -----	[Neutral gear] Minimum spindle speed in a tapping cycle [High gear]
0542	-----	Upper limit of the value output to the spindle motor
0543	-----	Lower limit of the value output to the spindle motor
0551	-----	Minimum spindle speed in the constant surface speed control mode (G96)
0556	-----	Maximum spindle speed for the constant surface speed control function
0576	-----	Permissible error in the number of error pulses between two spindles in the mode of (simple) synchronous control of the spindle
0577	-----	Spindle speed offset compensation
0585, 0586	-----	Gear-change point in S analog switching method B
0673 0677	----- -----	(Channel 1) Data for adjusting the gain in analog output (Channel 2)
0674 0678	----- -----	(Channel 1) Offset compensation in analog output (Channel 2)
0957 to 0959	-----	Limit of position error during spindle rotation at the maximum spindle speed
6780 to 6783	-----	Position loop gain of the servo axis in the Cs contour control mode
6784 to 6787 6788 to 6791 6792 to 6795 6796 to 6799	----- ----- ----- -----	(X) Position loop gain of the servo axis in the Cs contour control mode of each gear (Y) (Z) (4)
7516	-----	Data for adjusting the gain under sub-spindle control by the S command specified with four or five digits
7539	-----	Sub-spindle speed offset compensation

(14) Parameters related to tool compensation (1/1)

No.	Symbol	Description
0001#3	RS43	When a reset occurs, the vector of tool length compensation is cleared/is not cleared.
0003#6	TSLT	Tool length compensation is executed on the Z-axis (type A)/an axis vertical to the specified plane (type B).
0016#2	SUPM	In cutter compensation C, start-up and cancellation are executed through the method of type A/B.
0019#3	TLCD	Tool length compensation A or B/C
0030#2	OFCDH	Tool compensation memory C is disable/enabled.
0036#5	TLCDOK	In tool length compensation of type C, offset of two or more axes is disabled/enabled.
0036#6	OFRD	Tool length compensation and cutter compensation are specified with the H code./Tool length compensation and cutter compensation C are specified with the H code and D code, respectively.
0062#2	G40V	Operation by a single command (G40, G41, or G42)
0557	-----	Maximum travel distance that can be ignored on the outside of a corner in cutter compensation C

(15) Parameters related to canned cycles (1/2)

No.	Symbol	Description
0002#4, #5	PMXY1, 2	Axis and direction on and in which the tool is retracted in canned cycle G76 or G87
0011#4	MCF	When positioning of G81 terminates, the signal to specify an external operation function is not output/is output.
0012#4	FXCS	In canned cycle G74 or G84, reverse and forward spindle rotations are executed after M05 is output/even if M05 is not output.
0012#5	FXCO	In canned cycle G76 or G87, an oriented spindle stop is executed after M05 is output/even if M05 is not output.
0057#6	FXY	The drilling axis in a canned cycle is always the Z-axis/is a programmed axis.
0304	-----	M code for small-diameter peck drilling cycle
0305, 0306	-----	Spindle speed change ratio for small-diameter peck drilling cycle
0307, 0308	-----	Cutting feedrate change ratio for small-diameter peck drilling cycle
0309	-----	Minimum cutting feedrate ratio for small-diameter peck drilling cycle
0327	-----	Number of macro variable for counting number of retractions
0328	-----	Number of macro variable for counting number of times overload signal has been received
0398#6	CHGSKP	Using the overload torque signal, the feedrate and spindle speed are not changed/are changed.
0398#7	CHGNRM	Without using the overload torque signal, the feedrate and spindle speed are not changed/are changed.

9. PARAMETERS

For M series

(15) Parameters related to canned cycles (2/2)

No.	Symbol	Description
0403	-----	Clearance/cutting start point in a peck tapping cycle
0492	-----	Feedrate of retraction to the R position when address I is omitted
0493	-----	Feedrate of forward movement from the R position when address I is omitted
0531	-----	Clearance in canned cycle G73 (high-speed peck drilling cycle)
0532	-----	Cutting start point in canned cycle G83 (peck drilling cycle)
0941	-----	Clearance for small-diameter peck drilling cycle

(16) Parameters related to rigid tapping (1/2)

No.	Symbol	Description
0019#4	SRGTP	The rigid tapping selection signal (RGTP) is G123, #1/G135, #0.
0035#5	RGCTO	When the tool is retracted in rigid tapping, the time constant of acceleration/deceleration on the spindle and tapping axis is the same parameter as that for starting cutting/is a different parameter from that for starting cutting.
0037#6	VALT	In rigid tapping, the variable time constant switching function is not used/is used.
0040#2	RGTPPE	In rigid tapping, the rigid mode is canceled after the rigid DI signal is turned off/even before the rigid DI signal is turned off.
0063#3	VSLPC	An optional gear ratio between the spindle and position coder is not used/is used.
0063#4	RGDOV	When the tool is retracted, an override is disabled/enabled.
0065#5	TAPDRN	During tapping, a dry run is enabled/disabled.
0076#3	G84RGD	G84 and G74 are not handled as G codes for rigid tapping/are handled as G codes for rigid tapping.
0077#1	CT3G	Three-stage time constant switching is disabled/enabled.
0388#0	PCTPH	A peck tapping cycle is handled as a high-speed peck tapping cycle.
0388#2	RGMFH	A feed hold and single-block operation are enabled/disabled.
0388#3	RGORT	When rigid tapping is started, a spindle reference position return is not made/is made.
0388#5	SIG	At a gear-change, the SIND signal is disabled/enabled.
0388#6	CHKERC	During spindle rotation, the position error is checked according to the maximum spindle speed/specified spindle speed.
0254	-----	Type of acceleration/deceleration for the spindle and tapping axis
0255	-----	Spindle backlash amount in rigid tapping
0256	-----	M code for specifying the rigid tapping mode
0258	-----	Override value when the tool is retracted in rigid tapping
0378	-----	Override for rigid tapping return

(16) Parameters related to rigid tapping (2/2)

No.	Symbol	Description
0400 to 0402	-----	Time constant of acceleration/deceleration on the spindle and tapping axis when the tool is retracted
0613	-----	Time constant of acceleration/deceleration on the spindle and tapping axis
0614	-----	FL speed of exponential acceleration/deceleration on the spindle and tapping axis
0615	-----	Loop gain of position control on the spindle and tapping axis
0616	-----	(Low gear) Multiplier of loop gain on the spindle
0624	-----	(Neutral gear)
0625	-----	(High gear)
0617	-----	Allowable maximum spindle speed in rigid tapping
0618	-----	Effective area on the tapping axis in rigid tapping
0619	-----	Effective area on the spindle in rigid tapping
0620	-----	Limit of position error on the tapping axis during traveling
0621	-----	Limit of position error on the spindle during traveling
0622	-----	Limit of position error on the tapping axis under a stop
0623	-----	Limit of position error on the spindle under a stop
0626	-----	Feedrate for defining the reference lead in rigid tapping
0627	-----	Position error on the spindle in rigid tapping
0628	-----	Spindle pulse distribution in rigid tapping
0663 to 0665	-----	Number of teeth on the spindle when an optional gear ratio is selected
0666 to 0668	-----	Number of teeth on the position coder when an optional gear ratio is selected
0669 to 0671	-----	Loop gain of position control
0692	-----	(Low gear) Time constant of acceleration/deceleration on the spindle and tapping axis
0693	-----	(Neutral gear)
0694	-----	(Low gear) Maximum spindle speed in rigid tapping
0695	-----	(Neutral gear)
0696	-----	Instantaneous difference between errors on the spindle and tapping axis
0697	-----	Maximum difference between errors on the spindle and tapping axis
0799	-----	Integrated spindle pulse distribution in rigid tapping
0960	-----	Amount of return α for rigid tapping return

9. PARAMETERS

For M series

(17) Parameters related to scaling and coordinate system rotation (1/1)

No.	Symbol	Description
0036#0 to #2	SCL _z	Scaling for each axis is disabled/enabled.
0036#7	SRC	The unit of scaling magnification is 0.00001/0.001.
0041#0	RIN	An angle of coordinate system rotation is always specified in the absolute mode/is specified according to G90 or G91.
0063#6	ESCAL	Scaling and programmable mirror image on each axis are disabled/enabled.
0730	-----	Angle assumed when the angle of coordinate system rotation is not specified
0731 to 0733	-----	Magnification assumed when a scaling magnification is not specified

(18) Parameters related to unidirectional positioning (1/1)

No.	Symbol	Description
0029#0 to #3	G60 _z	In unidirectional positioning (G60) for each axis, the direction of approach is positive/negative.
0204 to 0207	-----	Distance of approach in unidirectional positioning for each axis

(19) Parameters related to control in the normal direction (1/1)

No.	Symbol	Description
0683	-----	Rotation speed around a controlled axis in the normal direction
0832	-----	Limit up to which an inserted rotation around a controlled axis in the normal direction can be ignored
0833	-----	Limit on travel distance that can be executed with the angle in the normal direction of the previous block

(20) Parameters related to index table indexing (1/1)

No.	Symbol	Description
0079#0	IXDDP	For the axis of index table indexing, the decimal point is input using the conventional method/pocket-calculator method.
0079#1	IXREL	On the screen, relative coordinates on the indexing axis which are not multiples of 360° are not rounded/are rounded.
0079#2	IXABS	On the screen, absolute coordinates on the indexing axis which are not multiples of 360° are not rounded/are rounded.
0079#3	IXINC	When the M code for rotation in the negative direction is not specified, the direction of rotation in the G90 mode minimizes the distance of rotation/does not minimize the distance of rotation.
0079#4	IXG90	An index command depends on specification of the G90 or G91 mode/always assumed that the G90 mode.
0079#7	IXTYP	Sequence of type A/type B
0249	-----	Code specifying a rotation in the negative direction
0839	-----	Minimum angle in index table indexing

(21) Parameters related to custom macro (1/1)

No.	Symbol	Description
0011#5	SBKM	By a macro statement, a single-block stop is not made/is made.
0040#1	DPOSUP	If data is output by the DPRINT command, leading zeros are output as space characters/as they are.
0040#5	TMCR	A T code is processed as the code of the tool function/code for calling O9000.
0040#6	COMC	When a reset occurs, common variables (#100 to #149) are made null/are not made null.
0040#7	LOCC	When a reset occurs, local variables (#01 to #33) are made null/are not made null.
0042	ASTCD	Hole pattern of the EIA code of an asterisk (*)
0043	EQCD	Hole pattern of the EIA code of an equal sign (=)
0044	SHPCD	Hole pattern of the EIA code of a sharp (#)
0053	LBLCD	Hole pattern of the EIA code of a square bracket (l)
0054	RBLCD	Hole pattern of the EIA code of a square bracket (r)
0056#1	MSKT	At an interrupt, absolute coordinates are not set as skip coordinates/are set as skip coordinates.
0056#2	MBLK	Custom macro interrupt of type I/type II
0056#3	MSTE	The interrupt signal uses the edge trigger method/status trigger method.
0056#4	MPRM	The M codes for enabling and disabling an interrupt are M96 and M97 respectively/are specified in parameters.
0056#5	MSUB	When an interrupt occurs, local variables are of the macro type/subprogram type.
0056#6	MCYL	During a cycle operation, a custom macro interrupt is disabled/enabled.
0056#7	MUSR	The custom macro interrupt function is disabled/enabled.
0057#4	CROUT	After data output in ISO code is completed with B/D PRINT, LF is output/CR/LF is output.
0220 to 0229	-----	G code for calling custom macro from O9010 to O9019
0230 to 0239	-----	M code for calling a custom macro from O9020 to O9029
0240 to 0242	-----	M code for calling a subprogram from O9001 to O9003
0246	-----	M code for enabling a custom macro interrupt
0247	-----	M code for disabling a custom macro interrupt
0248	-----	M code for calling a program registered in a file

(22) Parameters related to automatic tool length measurement (1/1)

No.	Symbol	Description
0558	-----	Feedrate during automatic tool length measurement
0813	-----	Value of γ in automatic tool length measurement
0814	-----	Value of ε in automatic tool length measurement

9. PARAMETERS

For M series

(23) Parameters related to graphic display (1/1)

No.	Symbol	Description
0046#5	GRPOS	On the screen of a solid drawing or tool path drawing, the current position is not displayed/is displayed.
0058#0	SGORG	If the coordinate system is changed during drawing, drawing is executed on the same coordinate system/on the assumption that the current drawing point is the current position set in the new coordinate system.
0058#1	SGTLC	During drawing of a solid drawing, tool length compensation is not executed/is executed.
0058#2	SG3PL	During drawing of a solid drawing, a triplane drawing is drawn with the first angle projection/ third angle projection.
0058#3	SGPLN	During drawing of a solid drawing, a top view is drawn without ridgelines/with ridgelines.
0058#4	SGFIN	During drawing of a solid drawing, a machining profile is drawn in the coarse mode/fine mode.
0058#5	SGCSR	On a tool path drawing, the current tool position or the center of a partial magnification is marked with ■ / × .
0253	-----	Change in the position of the cross section on a triplane drawing
0589 to 0592	-----	Margin on a solid drawing
0630, 0631	-----	Dot shift of 9-inch high-resolution graphic display

(24) Parameters related to the display of operation time and number of parts (1/1)

No.	Symbol	Description
0040#3	RWCNT	With M02 or M30, the total number of parts to be machined and the number of parts machined are counted/are not counted.
0219	-----	M code for counting the total number of parts to be machined and the number of parts machined
0600	-----	Number of parts required
0779	-----	Total number of parts to be machined

(25) Parameters related to tool life management (1/1)

No.	Symbol	Description
0039#0, #1	GST*	Number of tool groups that can be registered
0039#2	LCTM	The tool life is specified in terms of the number of times used/time period.
0039#3	TLSK	If the tool skip signal TLSKP (G140, #0) is input, the current group is skipped/the group number is input together.
0039#4	GRST	If the tool skip signal TLSKP (G140, #0) is input, the execution data of the current group is cleared/ the execution data of all groups is cleared.
0039#5	IGSK	The tool skip signal TLSKP (G140, #0) is accepted always/only while a tool under tool life management is used.
0039#6	IGIN	The tool back numbers are not ignored/are ignored.
0039#7	M6TCD	A T code in the block containing M06 is assumed as a back number/the command of the group to be selected next.
0041#7	M6TST	If a T code is specified in the block containing M06 under tool life management, the T code is processed as a back number or the number of the group to be selected next/the counting of the life of the tool group is started immediately.
0336	-----	M code for resuming the counting of the tool life
0599	-----	Number with which tool life management is ignored

(26) Parameters related to the position switch function (1/1)

No.	Symbol	Description
0310 to 0319	-----	Axis corresponding to the position switch
0840 to 0849	-----	Maximum value in the operating area of the position switch
0850 to 0859	-----	Minimum value in the operating area of the position switch

(27) Parameters related to manual handle feed/manual handle interrupt (1/2)

No.	Symbol	Description
0002#6	TJHD	During teaching in jog mode, the manual pulse generator is disabled/enabled.
0003#7	HLSE	If the manual pulse generators for three units are mounted, the axis selection signal is disabled/enabled.
0013#0	JHD	In the jog mode, the manual pulse generator is disable/enabled.
0018#0 to #3	N:MP2	For each axis, the magnification of handle feed (x 100) is enabled/disabled.
0019#0	MHPGB	Multihandle function of specification A/specification B
0060#4	HDLPM	If the handle of the manual pulse generator is rotated quickly, the reading and travel distance may not agree/the travel distance depends on the reading.

9. PARAMETERS

For M series

(27) Parameters related to manual handle feed/manual handle interrupt (2/2)

No.	Symbol	Description
0075#2	INHND	The travel increment of manual handle interrupt is set as the output unit and acceleration/deceleration is disabled/input unit and acceleration/deceleration is enabled.
0386#4 to #7	HDPIG _z	For each axis, the magnification of manual handle feed (x 1000) is enabled/disabled.
0117	-----	Manual handle pulse generator of the Z-axis or the fourth axis
0118	-----	Number of manual pulse generators used
0121	-----	Magnification of manual handle feed (M)
0699	-----	Magnification of manual handle feed (specification B)

(28) Parameters related to the software operator's panel (1/1)

No.	Symbol	Description
0017#0	OPG1	On the software operator's panel, the mode is not selected/is selected.
0017#1	OPG2	On the software operator's panel, the selection of a jog feed axis or rapid traverse is not executed/is executed.
0017#2	OPG3	On the software operator's panel, the selection of the manual pulse generator axis switch or magnification switch is not executed/is executed.
0017#3	OPG4	On the software operator's panel, jog feedrate override switch is not executed/is executed.
0017#4	OPG5	On the software operator's panel, BDT, SBK, MLK, or DRN switch is not executed/is executed.
0017#5	OPG6	On the software operator's panel, protect switch is not executed/is executed.
0017#6	OPG7	On the software operator's panel, feed hold switch is not executed/is executed.
0130 to 0137	-----	Axis and direction of jog feed corresponding to keys on the software operator's panel
0140 to 0203	-----	Character code of general-purpose switches on the software operator's panel

(29) Parameters related to program resumption (1/1)

No.	Symbol	Description
0387#0	SQDNC	During DNC operation, program resumption is disabled/enabled.
0124 to 0127	-----	Sequence of a jump to the program resumption point

(30) Parameters related to the high-speed cycle machining/high-speed remote buffer (1/1)

No.	Symbol	Description
0055#4 to #6	ITPCNT	Interval compensated for by G05 data
0597	-----	Number of controlled axes of the high-speed remote buffer. Maximum number of axes that can be simultaneously controlled in high-speed cycle machining

(31) Parameters related to PMC axis control (1/1)

No.	Symbol	Description
0024#0	IGNPMC	PMC axis control is enabled/disabled.
0030#0, #1	EAC*	Axis setting for executing PMC axis control (specification A)
0032#4 7032#4	PNGMLK PNGMLKS	On a PMC axis, a machine lock is enabled/disabled.
0032#6	EACSB	PMC axis control of specification A/specification B
0049#5	EFML10	Under PMC axis control, the specified feedrate (cutting feed) is multiplied by one/ten.
0052#0 to #7	NODIC*	Under PMC axis control, the current position display depends on the position of the decimal point with increment system 1 or 10/depends on the standard specifications.
0061#0 to #5	EBC*	Under PMC axis control (specification B), DI and DO used for each axis are of group A/B.
0062#6	AXPCF	To the actual speed display, traveling along a PMC controlled axis is added/is not added.
0063#5 7063#5	EAXOV EAXOVS	On a PMC axis, a dry run and override are disabled/enabled.
0066#3	EPMSKP	The skip signal used under PMC axis control is the same as the corresponding signal of the CNC/is a unique signal.
0066#6, #7	ERVF*	Magnification of the speed of feed per rotation under PMC axis control
0078#4	OVRIE	Under PMC axis control, the speed increases when the override signal is set to 0/1.
0078#6	RDRNE	Under PMC axis control, a dry run for the rapid traverse command is disable/enabled.
0078#7	EAXOVE	The dry run signal and override signal used under PMC axis control are the same as the corresponding signals of the CNC/are unique signals.
0387#7	EFERPD	Under PMC axis control, the parameter of the rapid traverse rate is the same as that of the CNC/ the rapid traverse rate is determined by the feedrate data specified with the axis control command.
0350	-----	Axis for which the velocity command is executed under PMC axis control
0462	-----	Time constant of linear acceleration/deceleration for the velocity specified with the velocity command
0657 to 0662 7657, 7658	----- -----	(X to 4) FL speed of exponential acceleration/ deceleration on a PMC axis during cutting feed (5, 6)
0672	-----	FL speed on a PMC controlled axis during reference position return
0685	-----	F0 speed of independent rapid traverse override on a PMC controlled axis
0698	-----	Maximum speed of feed per rotation about a PMC controlled axis
7021#6	SUB1	PMC controlled axes on the sub-CPU are the fifth and sixth axes/the fifth axis only.

9. PARAMETERS

For M series

(32) Parameters related to the surface grinding machine (slanted axis control) (1/1)

No.	Symbol	Description
0077#4	ZRTM1	When a manual reference position return is performed on the Y-axis, movement occurs on the Z-axis/does not occur on the Z-axis.
0077#5	AGLST	Slanted axis control is not executed/is executed.
0837	-----	Slant angle of the Y-axis
0838	-----	Minimum diameter of the grinding wheel when the diameter is checked

(33) Parameters related to simple synchronous control (1/1)

No.	Symbol	Description
0075#0, #1	SYNM _S	Master axis under simple synchronous control
0475	-----	Limit of difference between position errors of the master axis and slave axis

(34) Parameters related to the PMC (1/1)

No.	Symbol	Description
0028#1	PRCMSG	On the program check screen, the remaining travel distance is displayed/a message from the PMC is displayed.
0060#1	PCLDB	The baud rate during ladder loading is 4800 bps/9600 bps.
0070#6	PEXRD	The R or D area of the PMC-M is not extended/is extended.
0071#6	DPCRAM	With the PMC RAM board, PMC LOAD MENU is displayed/is not displayed.
0356 to 0359	-----	Number of characters that can be displayed in the remaining travel distance field on the program check screen
0476 to 0479	-----	First PMC address at which the characters in the remaining travel distance field are set

(35) Parameters related to the function for setting the zero point using the butt method (1/1)

No.	Symbol	Description
0360 to 0363	-----	Torque limit when the cycle for setting the zero point using the butt method is in progress
0942, 0943	-----	Butt speed when the cycle for setting the zero point using the butt method is executed
0944 to 0947	-----	Traveling speed when the zero point is set in the cycle for setting the zero point using the butt method
0948 to 0951	-----	Distance for setting the zero point when the cycle for setting the zero point using the butt method is executed

(36) Parameters related to the DNC (1/1)

No.	Symbol	Description
0390#7	NODC3	During DNC operation, reading is executed for each block/until the buffer is filled.
0320	-----	Time-out period of the no-response timer
0321	-----	Time-out period of the EOT signal timer
0323	-----	Maximum illegal data count
0324	-----	Maximum number of times a message is re-sent by NAK
0325	-----	Maximum number of characters that can be received after processing to stop transmission is completed
0347	-----	Format of connection between the CNC and the host on the DNC1 interface
0348	-----	Station address of the CNC on the DNC1 interface
0490	-----	Maximum length of the datagram (data section)

(37) Parameters related to the M-NET (1/1)

No.	Symbol	Description
0392#2	SRL1	The character length of the M-NET is 7 bits/8 bits.
0392#4	SRPE	The vertical parity check of the M-NET is not executed/is executed.
0392#5	SREP	The vertical parity of the M-NET is an odd parity/even parity.
0392#7	SRS2	The stop bit of the M-NET is bit 1/bit 2.
0259	-----	Byte length of DI of the M-NET
0260	-----	Byte length of DO of the M-NET
0261	-----	Station address of the M-NET
0262	-----	Baud rate of the M-NET
0458	-----	Timer monitor value after the sequence preparing for a connection to the same station is completed
0459	-----	Polling time monitor value in the normal sequence to the same station
0460	-----	Timer monitor value from the beginning of SAI transmission to the end of BCC transmission
0461	-----	Timer value from the end of reception to the beginning of transmission
0463	-----	Destination address of the spindle orientation signal

List of Parameters for each Function (for Lathe)

- (1) Parameters related to setting
- (2) Parameters related to the reader/punch interface/remote buffers
- (3) Parameters related to controlled axes/increment systems
- (4) Parameters related to coordinate systems
- (5) Parameters related to stroke limits
- (6) Parameters related to the feedrate
- (7) Parameters related to acceleration/deceleration control
- (8) Parameters related to the servo system
- (9) Parameters related to DI/DO
- (10) Parameters related to CRT/MDI, display, and editing
- (11) Parameters related to programs
- (12) Parameters related to pitch error compensation
- (13) Parameters related to spindle control
- (14) Parameters related to tool compensation
- (15) Parameters related to canned cycles
- (16) Parameters related to rigid tapping
- (17) Parameters related to custom macros
- (18) Parameters related to automatic tool compensation
- (19) Parameters related to graphic display
- (20) Parameters related to run time/parts count display
- (21) Parameters related to tool life management
- (22) Parameters related to the position switch function
- (23) Parameters related to manual handle feed/manual handle interrupt
- (24) Parameters related to the software operator's panel
- (25) Parameters related to restarting programs
- (26) Parameters related to high-speed cycle machining/high-speed remote buffers
- (27) Parameters related to PMC-based axis control
- (28) Parameters related to cylindrical grinding machines (GC)
- (29) Parameters related to the PMC
- (30) Parameters related to the butt-type reference position setting function
- (31) Parameters related to the DNC
- (32) Parameters related to the M-NET
- (33) Parameters related to the Cf-axis
- (34) Parameters related to 2-path lathes (TT)
- (35) Parameters related to polygon turning
- (36) Parameters related to synchronized control/axis recomposition

NOTE 1 In the explanation of each bit parameter, the left-hand side of a slash (/) indicates the state when the bit is set to 0, and the right-hand side of a slash indicates the state when the bit is set to 1.

Example: The current setting is 0/1.

NOTE 2 For details, see "Parameter Explanation."

(1) Parameters related to setting (1/1)

No.	Symbol	Description
0000	TVON	A TV check is not performed/is performed for program registration.
0000	ISO	EIA/ISO is used as the code system for program output.
0000	INCH	The metric/inch system is used as the least input increment (input unit) for programs.
0000	I/O	Input/output unit used via the reader/punch interface
0000	SEQ	Automatic sequence number insertion is not performed/is performed.
0000	PWE	Parameter write operation is disabled/enabled.
0000	TAPEF	Tape format is not converted/is converted.
0000	PRGNO	Program number for which sequence number reference is to be stopped
0000	SEQNO	Sequence number for which sequence number reference is to be stopped

(2) Parameters related to the reader/punch interface/remote buffers (1/2)

No.	Symbol	Description
0002#0 0012#0 0050#0 0051#0	STP2	(I/O=0) The number of stop bits is 1/2. (I/O=1) (I/O=2) (I/O=3)
0002#2 0012#2	ASR33	(I/O=0) FANUC PPR, etc./20-mA current interface (I/O=1)
0002#3 0012#3 0050#3 0051#3	RSASCI	(I/O=0) ISO or EIA/ASCII is used as the input code. (I/O=1) (I/O=2) (I/O=3)
0002#7 0012#7 0050#7 0051#7	NFED	(I/O=0) Line feed codes are output/not output before and after data output. (I/O=1) (I/O=2) (I/O=3)
0015#5	PRWD	The rewind signal is not output/is output to a portable tape reader.
0015#6	REP	If the same program number is already contained in memory when programs are registered via the reader/punch interface, an alarm is issued/the program is replaced.
0018#6	TVC	In a program comment, a TV check is not performed/is performed.
0038#1, #2 0038#4, #5 0038#6, #7	***, *** ***, *** ***, ***	(I/O=3) Input/output unit setting (I/O=2) (I/O=0,1)
0051#1	PARTY	(I/O=3) A parity bit is not provided/is provided.
0051#2	SYNAK	When protocol B is used, SYN and NAK code output control is not exercised/exercised.
0051#4	NCKCD	With the remote buffer interface, the CD (signal quality detection) state is monitored/not monitored.
0051#5	ECKL	An internal/external clock is used for the remote buffer baud rate clock.

9. PARAMETERS

For T series

(2) Parameters related to the reader/punch interface/remote buffers (2/2)

No.	Symbol	Description
0055#0	ASCII	ISO/ASCII code is used for communication of all data except NC data.
0055#1	ETX	The end code is CR/ETX.
0055#2	PROTCA	Communication protocol B/A is used.
0055#3	RS42	The remote buffer interface is RS-232-C/RS-422.
0055#7	RMSTS	When protocol A is used with the remote buffer interface, remote buffer state data in an SAT message is always sent back with no specification (=0)/a switching request in a SET message from the host is sent back unconditionally.
0070#7	ICR	In output using ISO code, the EOB code consists of an LF, CR, and CR/LF.
0075#7	IONUL	When a null character is included in EIA code being read, an alarm is not issued/is issued.
0076#5	PRWD	When reader/punch interface 2 is used, rewind is not performed/is performed.
0391#6	RS23BN	When the RS-232-C interface is used, the DC code is used/not used.
0399#6	FEDNUL	For feed operation in a significant information section, space/null characters are used.
0250 0251 0552 0553	----- ----- ----- -----	(I/O=2) Baud rate (I/O=3) (I/O=0) (I/O=1)

(3) Parameters related to controlled axes/increment systems (1/2)

No.	Symbol	Description
0001#0 7001#0	SCW SCWS	The metric/inch system is used for the least command increment (output unit) for linear axes.
0003#0 to #3 0066#4, #5 7003#0, #1	ZM* 	(X to 4) The reference position return direction and initial direction are positive/negative. (7, 8) (5, 6)
0019#2	XRC	For the X-axis, a diameter/radius is specified.
0024#1	LII10	The input/output unit is IS-B/IS-A.
0030#0, #4 0210, 0211 0285, 0286 7130, 7131	ADW*0 ----- ----- -----	(3, 4) Axis name (Display) (3, 4) (Command) (7, 8) (Display) (5, 6) (Display)
0032#2, #3 0062#4, #5 7032#0, #1	LIN* ADLIN* ROT*	The third and fourth axes are rotation axes/linear axes. The seventh and eighth axes are rotation axes/linear axes. (5, 6) Linear axis/rotation axis
0032#7 7032#7	ROT10 ROT10S	The parameter unit for inch output is 0.1 [deg/min]/1 [deg/min].
0049#4	S3JOG	The maximum number of simultaneously controlled axes in manual operation is 1/3.
0057#3	MIC	When the decimal point is omitted, the least input increment is not multiplied by 10/multiplied by 10.
0062#7	C8NG	The eighth axis is enabled/disabled.

(3) Parameters related to controlled axes/increment systems (2/2)

No.	Symbol	Description
0066#1	NBD78	When the number of controlled axes is greater than the number of servo axes, the extension of one controlled axis is disabled/enabled.
0069#4	BAX	As a command address for the third and fourth axes, B is not used/used.
0069#5, #6	***	Incremental or absolute command address for the third and fourth axes
0074#0 to #3	CRF*	If a command other than G28 specifies a movement by automatic operation when reference position return is not performed, an alarm is not issued/is issued.
0076#1	JZRN	The function for setting the reference position without dogs is disabled/enabled.
0385#6	RTLIN	Manual reference position return operation for a rotation axis is not performed/is performed linearly.
0388#1	ROAXC	The roll-over function for absolute coordinates is disabled/enabled.
0388#2	RODRC	When the ABS command is specified, the direction of rotation is the direction with the shortest distance/follows the sign of the specified value.
0388#3	ROCNT	The roll-over function for relative coordinates is disabled/enabled.
0388#5	IGPS90	P/S 090 is enabled/disabled.
0391#0 to #5	JZRN*	The function for setting the reference position without dogs is enabled/disabled.
0399#7	OUTZRN	If there is distance remaining to travel in manual reference position return, or a miscellaneous function is being executed, an alarm (P/S 091) is issued/not issued.
0269 to 0274	-----	Servo axis number
0279, 0280	-----	Attributes of the third and fourth axes
0281, 0282	-----	Axis synchronized with the X-/Z-axis or the third/fourth axis in synchronized control
0291, 0292	-----	Linear axis/rotation axis to which polar coordinate interpolation is applied

9. PARAMETERS

For T series

(4) Parameters related to coordinate systems (1/1)

No.	Symbol	Description
0002#1	PPD	Relative coordinates are not preset/are preset by coordinate system setting.
0010#6	WSFT	Workpiece coordinate system shift operation is not performed/is performed.
0010#7 7010#7	APRS APRSS	Automatic coordinate system setting is not performed/is performed in manual reference position return operation.
0028#5	RSFT	In direct input of a workpiece coordinate system shift amount, a diameter value/radius value is entered for the value of MX.
0075#6	CLCL	The local coordinate system is not canceled/is canceled in manual reference position return.
0388#6	WKZRST	In workpiece coordinate system setting, the workpiece coordinate system is not returned/is returned to G54 by an NC reset.
0388#7	WKZSFT	The workpiece shift and offset from the external workpiece reference point are not stored/are stored in the same memory location.
0708 to 711	-----	(X to 4) Reference position coordinates in automatic coordinate system setting
0815 to 0820 0825, 0826 7708, 7709	----- ----- -----	(X to 8) (7, 8) (5, 6)
0735 to 0738 0866, 0867	----- -----	(X to 4) Distance of the second reference position from the first reference position (7, 8)
0780 to 0783 0868, 0869	----- -----	(X to 4) Distance of the third reference position from the first reference position (7, 8)
0784 to 0787 0870, 0871	----- -----	(X to 4) Distance of the fourth reference position from the first reference position (7, 8)
0788	-----	Travel distance per rotation of a rotation axis
0940 to 0943	-----	Offset from the external workpiece reference point for each axis
0944 to 0947	-----	Offset from the first workpiece reference point for each axis (G54)
0948 to 0951	-----	Offset from the second workpiece reference point for each axis (G55)
0952 to 0955	-----	Offset from the third workpiece reference point for each axis (G56)
0956 to 0959	-----	Offset from the fourth workpiece reference point for each axis (G57)
0960 to 0963	-----	Offset from the fifth workpiece reference point for each axis (G58)
0964 to 0967	-----	Offset from the sixth workpiece reference point for each axis (G59)
7717	-----	Machine coordinate system when the fifth/sixth axis is a rotation axis

(5) Parameters related to stroke limits (1/1)

No.	Symbol	Description
0015#2	COTZ	The hardware overtravel signals (X018#5 *+LZ, X040#6 *-LZS) are valid/invalid.
0020#4	LM2	The switching signal for the second stored stroke limit is invalid/valid.
0024#4	INOUT	The out-of-range area of the third stored stroke limit is the inside/outside.
0065#3	PSOT	A stored stroke limit check is performed/is not performed until reference position return operation is performed.
0076#7	OTRFOM	An alarm is issued after/before a stroke limit is exceeded.
0700 to 0703	-----	(X to 4) First stored stroke limit in the positive direction for each axis
0821, 0822	-----	(7, 8)
7700, 7701	-----	(5, 6)
0704 to 0707	-----	(X to 4) First stored stroke limit in the negative direction for each axis
0823, 0824	-----	(7, 8)
7704, 7705	-----	(5, 6)
0747 to 0750	-----	Third stored stroke limit in the positive direction for each axis
0751 to 0754	-----	Third stored stroke limit in the negative direction for each axis
0760 to 0763	-----	Fourth stored stroke limit in the positive direction for each axis
0764 to 0767	-----	Fourth stored stroke limit in the negative direction for each axis
0770 to 0773	-----	Second stored stroke limit in the positive direction for each axis
0774 to 0777	-----	Second stored stroke limit in the negative direction for each axis

(6) Parameters related to the feedrate (1/2)

No.	Symbol	Description
0001#6	RDRN	For a rapid traverse command, dry run is disabled/enabled.
7001#6	RDRNS	
0008#4	MFPR	Manual synchronized feed (feed by spindle rotation) is disabled/enabled.
0008#5	ROVE	The rapid traverse override signal (G117#7 ROV2) is valid/invalid.
0010#0	ISOT	When no reference position is established, manual rapid traverse is disabled/enabled.
0015#3	SKPF	When G31 is specified, dry run, override, and automatic acceleration/deceleration are disabled/enabled.
0041#2	ZRNJF	Manual reference position return is not performed/performed at the jog feedrate.
0049#6	NPRV	When no position coder is used, the command for feed per rotation is disabled/enabled.
0049#7	FML10	The units of the parameters for setting the rapid traverse rate and cutting feedrate are not multiplied/are multiplied by 10.

9. PARAMETERS

For T series

(6) Parameters related to the feedrate (2/2)

No.	Symbol	Description
0059#0, #1	EDP*	For a plus-direction command for each axis, external deceleration is enabled only for rapid traverse/external deceleration is enabled for rapid traverse and cutting feed.
0059#3, #4	EDM*	For negative-direction commands for each axis, external deceleration is enabled only for rapid traverse/external deceleration is enabled for rapid traverse and cutting feed.
0065#5	G92ZAX	The parameters for a Z-axis time constant in threading and for an FL feedrate for acceleration/deceleration are common to all axes/Nos. 0627 and 0628.
0077#5	MICRF	The unit of the feedrate specified in F is 1 [mm/min]/0.001 [mm/min].
0393#5	STOV0	In rapid traverse, the tool is not stopped/stopped when a cutting feedrate override of 0% is specified.
0394#1	TFHOVR	The rapid traverse override for threading cycle retraction is the parameter-specified value/100 [%].
0397#2	OVR255	The 1%-step feedrate override signal is invalid/valid.
0399#2	PLAUT	During polar coordinate interpolation, if the specified rotation axis feedrate exceeds the maximum cutting feedrate, the command is output as is/the command is split.
0518 to 0521 0643, 0644 7518, 7519	----- ----- -----	(X to 4) Rapid traverse rate for each axis (7, 8) (5, 6)
0527	-----	Maximum allowable cutting feedrate
0528	-----	FL feedrate for acceleration/deceleration along the X-axis in threading (G92)
0530 7530	----- -----	(X to 4) FL feedrate for exponential acceleration/deceleration in cutting feed (5, 6)
0533 7533	----- -----	(X to 4) F0 feedrate for rapid traverse override (5, 6)
0534 7534	----- -----	(X to 4) FL feedrate for reference position return (5, 6)
0548	-----	FL feedrate for exponential acceleration/deceleration in manual feed
0559 to 0562	-----	Jog rapid traverse rate for each axis
0605 to 0608	-----	FL feedrate for exponential acceleration/deceleration in manual feed for each axis
0628	-----	FL feedrate for acceleration/deceleration along the Z-axis in threading (G92)
0636	-----	Feedrate for external deceleration
0657 to 0662 7657, 7658	----- -----	(X to 4) FL feedrate for exponential acceleration/deceleration in cutting feed for PMC axes (5, 6)

(7) Parameters related to acceleration/deceleration control(1/1)

No.	Symbol	Description
0020#5	NCIPS	An in-position check is performed/is not performed.
0045#3	LSUP	For acceleration/deceleration in cutting feed, exponential acceleration/deceleration/linear acceleration/deceleration after interpolation is used.
0050#4	SMZCT	Rapid traverse block overlap is invalid/valid.
0065#5	G92ZAX	The parameters for a Z-axis time constant in threading and for an FL feedrate for acceleration/deceleration are common to all axes/Nos. 0627 and 0628.
0522 to 0525	-----	(X to 4) Time constant for rapid traverse using linear acceleration/deceleration for each axis
0645, 0646 7522, 7523	----- -----	(7, 8) (5, 6)
0526	-----	Time constant for the X-axis in threading (G92)
0529	-----	Time constant for exponential acceleration/deceleration in cutting feed and manual feed
7529	-----	(5, 6)
0555	-----	Feedrate ratio at which the next block is started for rapid traverse block overlap
0601 to 0604	-----	(X to 4) Time constant for exponential acceleration/deceleration in manual feed for each axis
0627	-----	Time constant for the Z-axis in threading (G92)
0635	-----	Time constant for linear acceleration/deceleration after interpolation in cutting feed
0651 to 0656	-----	(X to 8) Time constant for exponential acceleration/deceleration in cutting feed for PMC axes
7651, 7652	-----	(5, 6)

(8) Parameters related to the servo system (1/3)

No.	Symbol	Description
0004 to 0007	GRD※	(X to 4) Size of a reference counter for each axis
0067, 0068 7004, 7005	GRD※ GRD※	(7, 8) (5, 6)
0570 to 0575 7570, 7511	----- -----	(X to 8) (5, 6)
0004 to 0007	DMR※	(X to 4) Detection multiplication factor (DMR) for each axis
0067, 0068 7004, 7005	DMR※ DMR※	(7, 8) (5, 6)
0010#2 7010#2	OFFVY OFFVYS	If VRDY=1 is set before PRDY=1, an alarm is issued/is not issued.
0021#0 to #5	APC※	(X to 8) For each axis, an absolute-position detector is not used/used.
7021#0, #1	APC※	(5, 6)
0021#6	NOFLUP	When the power is turned on before a reference position is established with an absolute-position detector, follow-up operation is performed/is not performed.

9. PARAMETERS

For T series

(8) Parameters related to the servo system (2/3)

No.	Symbol	Description
0022#0 to #5	ABS*	(X to 8) A reference position is not established/is already established with an absolute-position detector.
7022#0, #1	ABS*	(5, 6)
0035#7 7035#7	ACMR ACMRS	(X to 8) An arbitrary CMR is not used/is used. (5, 6)
0035#7 7035#7	ACMR ACMRS	(X to 8) An arbitrary CMR is not used/is used. (5, 6)
0037#0 to #5	SPTP*	(X to 8) As a position detector, a separate pulse coder is not used/is used.
7037#0, #1	SPTP*	(5, 6)
0037#7 7037#7	PLCO1 PLCO1S	(X to 8) A high-resolution pulse coder is not used/is used. (5, 6)
0041#1	THRDB	The threading start type is type A/type B.
0076#4	ADBLS	Cutting feed and rapid traverse separate backlash compensation is invalid/valid.
0389#3	TSKECR	When the torque limit skip function is used, the servo error present at the time of skipping is recovered/not recovered.
0390#0 to #5	NREQ*	When a reference position is not established upon power-up, an alarm is issued/is not issued to request reference position return when an absolute pulse coder is used for each axis.
0399#4	FUNO	When a servo alarm is detected by the CNC, follow-up operation is performed/is not performed.
0100 to 0103 0275, 0276 7100, 7101	----- ----- -----	(X to 4) Command multiplication factor (CMR) for each axis (7, 8) (5, 6)
0255	-----	Time interval for averaged current value display on the servo adjustment screen
0269 to 0274	-----	Servo axis number
0452 to 0457	-----	(X to 8 3RD WORD) Counter value at the APC reference position
0739 to 0742 0827, 0828 7580, 7581 7739, 7740	----- ----- ----- -----	(X to 4 LOWER 2 WORD) (7, 8 LOWER 2 WORD) (5, 6 3RD WORD) (5, 6 LOWER 2 WORD)
0500 to 0503 0637, 0638 7500, 7501	----- ----- -----	(X to 4) In-position width for each axis (7, 8) (5, 6)
0504 to 0507 0639, 0640 7504, 7505	----- ----- -----	(X to 4) Limit of position deviation during movement along each axis (7, 8) (5, 6)
0508 to 0511 0641, 0642 7508, 7509	----- ----- -----	(X to 4) Grid shift for each axis (7, 8) (5, 6)
0512 to 0515 0517 0632, 0633 7517	----- ----- ----- -----	(X to 4) Position control loop gain for each axis (X to 8) Position control loop gain common to all axes (7, 8) Position control loop gain for each axis (5, 6)

(8) Parameters related to the servo system (3/3)

No.	Symbol	Description
0535 to 0538 0647, 0648 7535, 7536	----- ----- -----	(X to 4) Backlash compensation for each axis (7, 8) (5, 6)
0593 to 0596 0649, 0650 7593, 7594	----- ----- -----	(X to 4) Limit of position deviation in the stop state for each axis (7, 8) (5, 6)
0673 to 0678	-----	(X to 4) Rapid traverse backlash compensation value for each axis

(9) Parameters related to DI/DO (1/1)

No.	Symbol	Description
0001#2	DCS	The signal from the START button on the MDI panel is routed/not routed via the machine.
0001#5 7001#5	DECI DECIS	(X to 4) The deceleration signal for reference position return starts deceleration when this bit is set to 0/1. (5, 6)
0003#4 7003#4	OVRI OVRIS	(X to 4) The override signal and rapid traverse override signal increase speed when this bit is set to 0/1. (5, 6)
0008#7	EILK	The start lock signal is common to all axes/de- fined for each axis.
0009#0 to #3 7009#0 to #3	TFIN TFIN	(X to 4) Acceptance time width for the M, S, or T function completion signal (G120#3 FIN) (5, 6)
0009#4 to #7 7009#4 to #7	TMF TMF	(X to 4) Time before the M, S, or T function code read signal is sent (5, 6)
0024#7	EDILK	The interlock signal for each axis direction is in- valid/valid.
0031#5	ADDCF	The PMC address for the GR1, GR2, and DRN signals is G0118/G0123.
0038#0	DEC34	The PMC addresses for the *DEC3 and *DEC4 signals are GX19#7 and X19#5/G16#7 and G17#7.
0041#3	ROVC	The rapid traverse override signals are ROV and ROV2/ROV1D, ROV2D, and ROV3D.
0045#2	RWDOUT	The rewind in-progress signal (F164#6 RWD) is output only while the tape reader is being re- wind/while programs in memory are being re- wind.
0045#7	HSIF	M, S, T, and B code processing is performed us- ing an ordinary/high-speed interface.
0070#4	DSTBGE	When output is started in background editing, the manual data input start signal (F150#5 DST) is output/is not output.
0394#6	POSILK	The start lock signal is invalid/valid for manual feed.
0113, 0114	-----	Position signal output grid number for each axis
0252	-----	Reset signal extension time
0713, 0714	-----	Position signal output grid width for each axis
0743 to 0746	-----	Distance to a contact surface for each axis

9. PARAMETERS

For T series

(10) Parameters related to CRT/MDI, display, and editing (1/3)

No.	Symbol	Description
0001#1	PROD	Tool length compensation data is included/is not included in relative coordinate display.
0002#1 7002#1	PPD PPDS	Relative coordinates are not preset/are preset by coordinate system setting.
0010#1	EBCL	When programs stored in memory are displayed, the EOB code is displayed using ;/*.
0011#7	MCINP	Programs are not stored/are stored in memory with G117#0 MINP.
0013#4	INHMN	Menu display is performed/not performed.
0014#2	STDP	The actual speed of the spindle, S codes, and T codes are not displayed/are displayed.
0015#0	CBLNK	The cursor blinks/does not blink.
0015#1	NWCH	When tool wear compensation values are displayed, the character W is displayed/is not displayed.
0015#6	REP	If the same program number is already contained in memory when programs are registered via the reader/punch interface, an alarm is issued/the program is replaced.
0018#7	EDITB	The type of editing using the standard keyboard is edit operation A/B.
0019#6	NEOP	When a program is registered, M02, M30, or M99 ends/does not end registration.
0019#7	DBCD	When the diagnostic screen is displayed, timer and counter data are displayed in binary/decimal.
0023#0 to #6	*:*:*	Display language setting
0028#0	PRCPOS	On the program check screen, relative/absolute coordinates are displayed.
0028#2	DACTF	Actual speed is not displayed/is displayed.
0028#3	MMDL	In MDI operation B, the modal state is not displayed/is displayed.
0029#0, #1	DSP*	The current position of the third/fourth axis is not displayed/is displayed.
0029#6	DSPSUB	The fifth/sixth axis is not displayed/is displayed on the absolute/relative position display screen.
0038#3	FLKY	The keyboard of the CRT unit or MDI panel employs the standard keys/full keypad.
0040#0	NAMPR	Program names are not displayed/are displayed in the program directory.
0040#4	SORT	The program directory is displayed in the order of program registration/in the ascending order of program numbers.
0045#0	RDL	During external control of input/output units, read operation follows the specification of No. 0015#6 REP/registration is performed after deletion of all programs.
0045#1	RAL	In reading via the reader/punch interface, all programs are registered/only the first program is registered.
0048#7	SFFDSP	Soft keys are displayed and controlled according to the additional option function specification/regardless of the additional option function specification.

(10) Parameters related to CRT/MDI, display, and editing (2/3)

No.	Symbol	Description
0050#1	NOFMK	In sequence number search, a format check is performed/is not performed.
0056#0	NOCND	When the part program storage length is 120 m/320 m, or when the background edit function is provided, automatic memory compression is performed/is not performed.
0057#0	PWNNC	The MMC/CNC screen is displayed when the power is turned on.
0057#1	ALMNC	When an alarm is issued on the CNC during MMC screen display, the MMC/CNC screen is displayed.
0057#2	SALNC	When a system alarm is issued on the CNC during MMC screen display, the MMC/CNC screen is displayed.
0060#0	DADRD	Addresses are not displayed/are displayed on the diagnostic screen.
0060#2	LDDSPG	Dynamic ladder display is not performed/is performed.
0060#3	C14DGN	When the 14-inch CRT unit is used, the diagnostic screen is displayed in the 9-inch/14-inch format.
0060#5	OPMNDP	Operating monitor display is invalid/valid.
0060#6	EXTSP	The display and search operations are disabled/enabled for protected programs.
0060#7	IGNCOL	When the 9-inch high-resolution CRT unit is used, a monochrome/color display is used.
0061#6	DSP78	For current position display using absolute and relative coordinates, the seventh and eighth axes are not displayed/are displayed.
0063#0	MTDSP1	Output system/input system for machine coordinate display
0063#1	PRSTIN	For automatic coordinate system setting based on inch input, No. 0708 and up/No. 0815 and up are used.
0064#0	SETREL	For relative coordinate presetting, the relative coordinates are cleared to 0/preset to arbitrary values.
0064#1	ALLPRE	For relative coordinate presetting, the standard specification is used/axis selection is performed using numeric keys.
0064#3	IOMDGN	On the diagnostic screen, I/O link and extended R/D area DI/DO display are not provided/are provided.
0064#5	NPA	When an alarm is issued or an operator message is entered, the screen display switches/does not switch to the alarm screen or message screen.
0066#0	ALL56	On the overall current position screen, the fifth and sixth axes are not displayed/are displayed.
0076#2	IOP	NC program input and output operations can be stopped by resetting the NC/only by selecting the [STOP] soft key.
0077#2	SGP	Servo waveform display is invalid/valid.
0077#6	HLKEY	MDI key processing is/is not high-speed.
0078#0	NOINOW	Input of a tool wear compensation value through the MDI keys is not prohibited/is prohibited.

9. PARAMETERS

For T series

(10) Parameters related to CRT/MDI, display, and editing (3/3)

No.	Symbol	Description
0078#1	NOINOG	Input of a tool geometry compensation value through the MDI keys is not prohibited/is prohibited.
0078#2	NOINMV	Input of macro variables through the MDI keys is not prohibited/is prohibited.
0078#3	NOINWZ	Input of an offset from the workpiece reference point through the MDI keys is not prohibited/is prohibited.
0389#0	SRVSET	The servo setting screen is displayed/is not displayed.
0393#2	WKNMDI	In the automatic operation activation state/automatic operation stop state, input of an offset from the workpiece reference point through the MDI keys is not prohibited/is prohibited.
0393#7	DGNWEB	When PWE=0, PMC parameter input is prohibited/is not prohibited.
0394#7	CAKEY	On the parameter, diagnostic, and offset screens, one-character cancellation using the CAN key is disabled/enabled.
0395#1	TLSCUR	On the offset screen, the cursor position is not preserved/is preserved.
0397#7	SERNAL	The details of alarm 409 are not displayed/are displayed.
0257 to 0263	-----	Tool post name (first to seventh characters) displayed on the screen
0264 to 0267	-----	Axis name subscript for each axis
0337 to 0346	-----	Title character code
0351 to 0355	-----	NC name character code
0550	-----	Incremental value for automatic sequence number insertion
0797	-----	Password
0798	-----	Key

(11) Parameters related to programs (1/2)

No.	Symbol	Description
0010#4	PRG9	The editing of subprograms O9000 to O9999 is not prohibited/is prohibited.
0011#6	G01	Upon power-up, the G00/G01 mode is set.
0015#7	CPRD	When the decimal point is omitted, the least input increment/mm, inch, deg, or sec unit is used.
0016#3	NPRD	The decimal point input and display function is used/is not used.
0016#5	SPAG	An angle for direct drawing dimension programming is specified by angle/supplementary angle.
0019#5	M02NR	After M02 is executed, control returns/does not return to the start of the program.
0028#4	EXTS	External workpiece number search/external program number search
0029#4	ADRC	As a chamfering or corner R address, C or R/, C or ,R is used.
0030#7	G91	Upon power-up, the G90/G91 mode is set.
0032#5	BLIN	For B 8-digit input, 1.000 is always used/1.000 (millimeter) or 1.0000 (inch) is used.

(11) Parameters related to programs (2/2)

No.	Symbol	Description
0036#1	GSP	The G code system is of type A/type B.
0036#5	GSPC	The G code system is of type B (A)/type C.
0040#2	MAPS	Address Q for the pass point signal output function cannot be used/can be used.
0040#5	TMCR	The T code is for a tool function/for calling O9000.
0045#6	CLER	The reset button, external reset signal, and emergency stop operation set the reset state/clear state.
0057#7	MBCLR	Upon reset, MDI-B programs are not cleared/are cleared.
0063#2	M198P	M198 address P specifies a file number/program number.
0065#7	M3B	One block can contain only one M code/up to three M codes.
0389#2	PRG8	The editing of subprograms O8000 to O8999 is not prohibited/is prohibited.
0391#7	NOCLR	When the clear state is entered, certain G codes are cleared/are not cleared.
0393#3	M3RQNG	The specification of a three-digit M code is valid/invalid.
0393#6	RADCHK	When circular interpolation is specified, the difference between the radius at the start point and the radius at the end point is not checked/is checked.
0395#0	DLG99	In the feed per rotation mode, a dwell command is specified by time/spindle speed.
0396#7	EORRE	If an EOR is read without reading a program end code, an alarm is issued/the reset state is set.
0111, 0112	-----	M code which performs no buffering
0243	-----	Third digit (hundreds digit) of a wait M code
0248	-----	M code for calling programs stored in a file
0554	-----	Program number for parameters/macro variables/diagnostic data
0587	-----	M code for specifying spindle orientation
0588	-----	M code for canceling spindle indexing
0589	-----	Initial value of an M code used for spindle indexing
0876	-----	Arc radius error limit

9

(12) Parameters related to pitch error compensation (1/1)

No.	Symbol	Description
0011#0, #1	PML*	(X to 4) Pitch error compensation magnification
7011#0, #1	PML*S	(5, 6)
0756 to 0759	-----	(X to 4) Compensation interval in pitch error compensation for each axis
7713, 7714	-----	(5, 6)
1000 to 6000	-----	Zero position for pitch error compensation for each axis
1001 to 6128	-----	Pitch error compensation for each axis

9. PARAMETERS

For T series

(13) Parameters related to spindle control (1/3)

No.	Symbol	Description
0003#6, #7	PSG※	Gear ratio between the spindle and position coder
0064#6, #7	P2SG※	Gear ratio between the spindle and second position coder
0013#5	ORCW	In orientation, S analog output is positive/negative.
0013#6, #7	TCW, CWM	Sign of S analog output
0024#2	SCTO	The spindle speed arrival signal (G120#4 SAR) is not checked/is checked.
0028#7	PNOSMP	The number of sampling operations for finding the average spindle speed is 4/1.
0031#0	SNRL	Upon reset, the spindle indexing mode is not canceled/is canceled.
0031#1	SDRT	The direction of spindle indexing with an M code is positive/negative.
0049#0	EVSF	When an S code is specified, the S code and SF are not output/are output at all times.
0049#2	CHKSP	When SIND=1, spindle speed fluctuation detection is disabled/enabled.
0062#3	SPMRPM	The unit of parameter data for spindle speed control is 1/10 [RPM].
0065#1	CZRN	The first G00 command after the mode is switched to the Cs contour control mode performs positioning after performing a reference position return/performs ordinary positioning.
0070#0, #1	DAC※	For D/A conversion, the analog interface board is not used/used.
0070#3	MSPDB	The multi-spindle control method is type A/type B.
0071#0	ISRLPC	When a serial interface spindle is used, the position coder signal is received through an optical fiber/M27 connector.
0071#1	HISSC	The sampling time for constant surface speed control remains unchanged from the conventional value/is the conventional value divided by 4.
0071#2	SRLMSP	In multi-spindle control, the second axis is analog/serial.
0071#3	SSMPC	M27/the second spindle control unit is used for the second position coder.
0071#4	SRL2SP	One/two serial interface spindles is/are connected.
0071#7	FSRSP	Serial interface spindles are not used/are used.
0074#7	PLCREV	A feedback pulse signal from a position coder represents an absolute value/is signed.
0080#2, #3	MORCM※	For the first and second spindle motors, the spindle orientation function with the stop position set externally is not used/is used.
0080#6, #7	SP※NEG	During spindle synchronization, the rotation directions of the first and second spindle motors are the same as the specified sign/opposite to the specified sign.
0108	-----	Spindle speed when the spindle rotates at a constant speed
0110	-----	Delay timer used to check the spindle speed arrival signal (G120#4 SAR)

(13) Parameters related to spindle control (2/3)

No.	Symbol	Description
0249	-----	Magnification for spindle speed ratio control over serial interface spindles
0303	-----	Permissible pulse error for phase synchronization in the synchronized control mode
0381#7	IGNSIC	A serial/analog spindle is connected to tool post 2.
0397#0	SPLME	The permissible rate and fluctuation rate of the spindle speed fluctuation detection function are 1 [%]/0.1 [%].
0516	-----	Data for gain adjustment in constant surface speed control (analog output)
0531	-----	Permissible rate (Q) used in spindle speed fluctuation detection for determining whether a specified speed is reached
0532	-----	Spindle speed fluctuation rate (R) used in spindle speed fluctuation detection for issuing an alarm
0539	-----	Spindle speed offset value
0540 to 0543	-----	Speed when the spindle speed command voltage for each gear is 10 [V]
0546	-----	Value for compensating for drift that occurred in a Cs-axis servo loop
0551	-----	Minimum spindle speed in the constant surface speed control mode (G96)
0556	-----	Maximum spindle speed for the constant surface speed control option
0564	-----	Permissible speed fluctuation that does not cause an alarm to be issued in spindle speed fluctuation detection
0576	-----	Permissible pulse error between two spindles in the spindle (simple) synchronized control mode
0580 to 0583	-----	Spindle servo loop gain multiplier in Cs-axis control for each gear
0584	-----	Spindle loop gain
0585	-----	F0 feedrate for spindle rapid traverse override
0586	-----	FL feedrate for spindle orientation
0587	-----	M code for specifying spindle orientation
0588	-----	M code for canceling spindle indexing
0589	-----	Initial value of an M code used for spindle indexing
0590	-----	Basic angular displacement for spindle indexing using an M code
0613	-----	Data for second spindle gain adjustment in constant surface speed control
0614	-----	Spindle speed offset value for the second spindle
0615, 0616	-----	Speed when the second spindle speed command voltage for each gear is 10 [V]
0617	-----	Data for third spindle gain adjustment in constant surface speed control
0618	-----	Spindle speed offset value for the third spindle
0619, 0620	-----	Speed when the third spindle speed command voltage for each gear is 10 [V]
0712	-----	Time before a spindle speed fluctuation check is started

9. PARAMETERS

For T series

(13) Parameters related to spindle control (3/3)

No.	Symbol	Description
6780 to 6783	-----	Position loop gain of a servo axis in the Cs contour control mode
6784 to 6787	-----	(X) Position loop gain of a servo axis in the Cs contour control mode for each gear
6788 to 6791	-----	(Y)
6792 to 6795	-----	(Z)
6796 to 6799	-----	(4)
7516	-----	Data for gain adjustment in subspindle S4/S5 digit control
7539	-----	Subspindle speed offset value

(14) Parameters related to tool compensation (1/2)

No.	Symbol	Description
0001#3	TOC	In the reset state, offset vectors are not canceled/are canceled.
0001#4	ORC	For offset values, a diameter value/radius value is specified.
0008#6	NOFC	Offset values are loaded/are not loaded into a counter.
0010#5	DOFSI	Direct input of tool offset values is not performed/is performed.
0013#1	GOFU2	A tool geometry compensation number is specified using the least significant/most significant digit of a T code.
0013#2	GMOFS	Tool geometry compensation is performed by shifting the coordinate system/by moving the tool.
0013#3	GOFC	Tool geometry compensation is not canceled/is canceled by position number 0.
0014#0	T2D	A T code is specified using 4 digits/2 digits.
0014#1	GMCL	Tool geometry compensation is not canceled/is canceled when the reset state is entered.
0014#4	OFSB	Tool offsetting is performed in a block containing a T code/performed together with axis movement.
0014#5	WIGA	A limit is not imposed/is imposed on tool offset setting.
0014#6	T2T4	When a T code is specified using 2 digits, the upper 2 digits are assumed to be 00/are assumed to be the same as the lower 2 digits.
0015#4	MORB	In direct input of measurement values, the record button is not used/is used.
0024#6	QNI	When tool compensation measurement direct input function B is used, a tool compensation number is not selected/is selected automatically in the offset write mode.
0036#3	YOF3	Y-axis offset is used with the fourth axis/third axis.
0075#3	WNPT	A virtual tool tip number for tool-tip radius compensation is specified using a geometric/wear compensation number.
0122	-----	Tool compensation number when the tool compensation measurement value direct input B function is used

(14) Parameters related to tool compensation (2/2)

No.	Symbol	Description
0395#4	ADDLA	During tool-tip radius compensation, if two blocks specifying no movement are specified consecutively, the offset vector is assumed to be the vector perpendicular to the movement performed in the previous block at the end point/the intersection vector.
0557	-----	Maximum ignorable travel distance along the outside of a corner in tool-tip radius compensation
0728	-----	Maximum tool wear compensation value in incremental input
0729	-----	Maximum tool wear compensation value

(15) Parameters related to canned cycles (1/1)

No.	Symbol	Description
0031#2	TAPM	In a tapping cycle, M03 or M04/M05 is output in the case of forward or reverse rotation.
0031#3	ILVL	In a drilling cycle, the initial point is not updated/is updated when a reset occurs.
0031#4	RTCT	The peck drilling cycle is type A/B.
0393#1	MCQSCH	In a multiple repetitive canned cycle for lathes, a sequence number check with Q specified is not performed/is performed.
0393#4	CHKMRC	In a multiple repetitive canned cycle for lathes, specification of a pocket figure is valid/invalid.
0109	-----	Cut width in threading cycle G92/G76
0204	-----	M code for C-axis clamping in a drilling canned cycle
0212, 0213	-----	M code for forward spindle rotation/reverse spindle rotation in a drilling canned cycle
0591	-----	Dwell time used when C-axis unclamping is specified in a drilling cycle
0592	-----	Retract dimension (d) used for peck drilling in a drilling cycle
0717, 0718	-----	Depth of cut/retract dimension in the multiple repetitive canned cycles (G71, G72)
0719, 0720	-----	Retract dimension for each axis in the multiple repetitive canned cycle (G73)
0721	-----	Number of divisions in the multiple repetitive canned cycle (G73)
0722	-----	Return distance in the multiple repetitive canned cycles (G74, G75)
0723	-----	Number of times finishing is performed in the multiple repetitive canned cycle (G76)
0724	-----	Tool angle in the multiple repetitive canned cycle (G76)
0725	-----	Minimum depth of cut in the multiple repetitive canned cycle (G76)
0726	-----	Finishing allowance in the multiple repetitive canned cycle (G76)

9. PARAMETERS

For T series

(16) Parameters related to rigid tapping (1/1)

No.	Symbol	Description
0029#3	RGCTO	In rigid tapping, the time constant for acceleration/deceleration of the spindle and tapping axis during withdraw are specified using the same parameter as for cutting/using a separate parameter.
0041#0	RGTPPE	In rigid tapping, the rigid mode is canceled after/before the rigid DI signal is turned off.
0063#4	RGDOV	Withdraw operation overriding is disabled/enabled.
0063#6	VSLPC	An arbitrary gear ratio is not used/is used between the spindle and position coder.
0076#3	G84RGD	G84 and G74 are not used/are used as G codes for rigid tapping.
0397#3	RGMFH	The feed hold and single block functions are enabled/disabled.
0399#5	SIG	In gear switching, the SIND signal is invalid/valid.
0214 to 0217	-----	Spindle backlash distance in rigid tapping
0253	-----	M code for specifying the rigid tapping mode
0254	-----	Override value for withdraw in rigid tapping
0400	-----	Tapping axis in-position width in rigid tapping
0401	-----	Spindle in-position width in rigid tapping
0402	-----	Limit on position deviation during movement along the tapping axis
0403	-----	Limit on position deviation during spindle movement
0404	-----	Limit on position deviation while movement along the tapping axis is stopped
0405	-----	Limit on position deviation while spindle movement is stopped
0406 to 0410	-----	Loop gain for position control over the spindle and tapping axis (for each gear)
0411 to 0414	-----	Spindle loop gain multiplier
0415 to 0418	-----	Time constant for acceleration/deceleration for the spindle and tapping axis
0419 to 0422	-----	Time constant for acceleration/deceleration for the spindle and tapping axis during withdraw
0423 to 0426	-----	Maximum allowable spindle speed during rigid tapping
0427 to 0430	-----	Number of teeth on the spindle side when an arbitrary gear ratio is used
0431 to 0434	-----	Number of teeth on the position coder side when an arbitrary gear ratio is used
0435	-----	Position deviation of the spindle during rigid tapping
0436	-----	Amount of pulse distribution to the spindle during rigid tapping
0473	-----	Momentary error difference between the spindle and tapping axis
0483	-----	Maximum error difference between the spindle and tapping axis

(17) Parameters related to custom macros (1/1)

No.	Symbol	Description
0011#5	SBKM	A macro statement does not cause/causes a single block stop.
0040#1	DPOSUP	When data is output with the DPRINT command, spaces are output/no particular operation is performed for leading zeros.
0040#5	TMCR	The T code is for a tool function/ for calling O9000.
0040#6	COMC	Upon reset, common variables (#100 to #149) are placed/not placed in the null state.
0040#7	LOCC	Upon reset, local variables (#01 to #33) are placed/not placed in the null state.
0042	ASTCD	Hole pattern corresponding to EIA code ∗:
0043	EQCD	Hole pattern corresponding to EIA code =
0044	SHPCD	Hole pattern corresponding to EIA code #
0053	LBLCD	Hole pattern corresponding to EIA code [
0054	RBLCD	Hole pattern corresponding to EIA code]
0056#1	MSKT	When an interrupt is generated, absolute coordinates are not set/are set as skip coordinates.
0056#2	MBLK	Custom macro interrupt type I/II
0056#3	MSTE	The interrupt signal is based on the edge trigger method/status trigger method.
0056#4	MPRM	M codes for enabling and disabling an interrupt are M96 and M97 respectively/determined by parameter setting.
0056#5	MSUB	The local variables used for an interrupt are macro type/subprogram type.
0056#6	MCYL	During cycle operation, the custom macro interrupt function is disabled/enabled.
0056#7	MUSR	The custom macro interrupt function is disabled/enabled.
0057#4	CROUT	Upon completion of ISO code data output by B/D PRINT, an LF code/CR and LF codes are output.
0218	-----	Number of macro variables common to tool posts
0220 to 0229	-----	G code for calling custom macros O9010 to O9019
0230 to 0239	-----	M code for calling custom macros O9020 to O9029
0240 to 0242	-----	M code for calling subprograms O9001 to O9003
0246	-----	M code for enabling custom macro interrupts
0247	-----	M code for disabling custom macro interrupts
0248	-----	M code for calling programs stored in a file

9

(18) Parameters related to automatic tool compensation (1/1)

No.	Symbol	Description
0558	-----	Feedrate used for measurement in automatic tool compensation
0731, 0732	-----	Value of γ in automatic tool compensation
0733, 0734	-----	Value of ε in automatic tool compensation

9. PARAMETERS

For T series

(19) Parameters related to graphic display (1/1)

No.	Symbol	Description
0024#3	GNSR	The actual position with tool-tip radius compensation reflected is drawn/the programmed position is drawn.
0046#5	GRPOS	On the graphic display screen, the current position is not displayed/is displayed.
0123	-----	Drawing coordinate system used with the graphic function
0630, 0631	-----	Dot shift amount for 9-inch high-resolution graphic display

(20) Parameters related to run time/parts count display (1/1)

No.	Symbol	Description
0040#3	RWCNT	With M02 or M30, the total number of machined parts and the number of machined parts is counted/is not counted.
0219	-----	M code for counting the total number of machined parts and the number of machined parts
0600	-----	Number of required parts
0779	-----	Total number of machined parts

(21) Parameters related to tool life management (1/1)

No.	Symbol	Description
0039#0, #1	GST※	Number of groups that can be registered in tool group setting
0039#2	LCTM	Tool life is specified by a count/time.
0039#3	TLSK	When the tool skip signal (G140#0 TLSKP) is applied, the group currently selected is skipped/the group number is also entered.
0039#4	GRST	When the tool skip signal (G140#0 TLSKP) is applied, the execution data of the group/execution data of all groups is cleared.
0039#5	IGSK	The tool skip signal (G140#0 TLSKP) is accepted always/only when a tool subject to tool life management is being used.
0394#0	TLCSAM	When multiple offset numbers are involved, counting is performed for each identical tool number/each tool.
0394#5	NOWOFS	@ is prefixed/is not prefixed to a compensation number subject to tool life management.
0117	-----	Offset number compensation value used with the simplified tool life management function
0118	-----	Tool selection number compensation value used with the simplified tool life management function
0119	-----	Maximum offset number used with the simplified tool life management function
0120	-----	Maximum tool selection number used with the simplified tool life management function
0336	-----	M code for restarting tool life count operation
0599	-----	Tool life management ignore number

(22) Parameters related to the position switch function (1/1)

No.	Symbol	Description
0310 to 0319	-----	Axis corresponding to a position switch
0840 to 0849	-----	Maximum position switch operation range
0850 to 0859	-----	Minimum position switch operation range

(23) Parameters related to manual handle feed/manual handle interrupt (1/1)

No.	Symbol	Description
0002#5	HSLE	When two manual pulse generators are installed, the axis selection signal is invalid/valid.
0002#6	TJHD	In the TEACH IN JOG mode, a manual pulse generator is disabled/enabled.
0013#0	JHD	In the JOG mode, a manual pulse generator is disabled/enabled.
0075#2	INHND	The unit of manual handle interrupt movement is the output unit with acceleration/deceleration disabled/is the input unit with the acceleration/deceleration enabled.
0077#4	HDLPM	When the handle of a manual pulse generator is turned quickly, the scale indication and travel distance may not match/the scale indication and travel distance match.
0386#0 to #3	HPNEG*	For each axis, the direction of manual handle feed is the same/opposite.
0386#4 to #7	HDPIG*	For each axis, the magnification (x 10000) of manual handle feed is used/is not used.
0121	-----	Manual handle feed magnification (M)
0699	-----	Manual handle feed magnification (specification B)

(24) Parameters related to the software operator's panel (1/1)

No.	Symbol	Description
0017#0	OPG1	On the software operator's panel, mode selection is not performed/is performed.
0017#1	OPG2	On the software operator's panel, jog feed axis selection is not performed/is performed.
0017#2	OPG3	On the software operator's panel, manual pulse generator axis selection and magnification switching are not performed/are performed.
0017#3	OPG4	On the software operator's panel, jog feedrate override switching is not performed/is performed.
0017#4	OPG5	On the software operator's panel, BDT, SBK, MLK, and DRN switching is not performed/is performed.
0017#5	OPG6	On the software operator's panel, protect switching is not performed/is performed.
0017#6	OPG7	On the software operator's panel, feed hold switching is not performed/is performed.
0130 to 0137	-----	Jog feed axes and directions of the keys on the software operator's panel
0140 to 0203	-----	Character codes of the general-purpose switches on the software operator's panel

9. PARAMETERS

For T series

(25) Parameters related to restarting programs (1/1)

No.	Symbol	Description
0387#0	SQDNC	In DNC operation, program restarting is disabled/enabled.
0124 to 0127	-----	Order of movement to program restart positions

(26) Parameters related to high-speed cycle machining/high-speed remote buffers (1/1)

No.	Symbol	Description
0055#4 to #6	ITPCNT	G05 data interpolation interval
0597	-----	Number of high-speed remote buffer controlled axes

(27) Parameters related to PMC-based axis control (1/2)

No.	Symbol	Description
0024#0	IGNPMC	PMC-based axis control is enabled/disabled.
0032#4 7032#4	PNGMLK PNGMLKS	Machine locking along a PMC axis is enabled/disabled.
0032#6	EACSB	PMC axis control is based on specification A/B.
0049#5	EFML10	A feedrate command (cutting feed) for PMC axis control is multiplied by 1/10.
0052#0 to #7	NODIC※	Current position display for PMC axis control follows the decimal point position based on the setting unit 1/10/follows the standard specifications.
0061#0 to #5	EBC※	For each axis, group A/B of the DI and DO signals for PMC axis control (specification B) is used.
0062#6	AXPCF	Movement along a PMC controlled axis is added/is not added to actual speed display.
0063#5 7063#5	EAXOV EAXOVS	For PMC axes, the dry run and override functions are disabled/enabled.
0066#3	EPMSKP	In PMC-based axis control, the same skip signal used with the CNC/a separate signal is used.
0066#6, #7	ERVF※	Magnification of a feedrate for feed-per-rotation in PMC axis control
0078#4	OVRIE	In PMC axis control, the override signal increases speed when this bit is set to 0/1.
0078#6	RDRNE	In PMC axis control, the dry run function cannot/can be used with a rapid traverse command.
0078#7	EAXOVE	In PMC axis control, the dry run and override signals are the same as those used with the CNC/separate dry run and override signals are used.
0387#7	EFERPD	A rapid traverse rate in PMC axis control is specified by the same parameter as with the CNC/specified by the feedrate data of an axis control command.
0350	-----	Axis for which a feedrate is specified in PMC axis control
0651 to 0656 7651, 7652	-----	(X to 8) Time constant for exponential acceleration/deceleration during cutting feed for a PMC axis (5, 6)
0657 to 0662 7657, 7658	-----	(X to 8) FL feedrate for exponential acceleration/deceleration during cutting feed for a PMC axis (5, 6)

(27) Parameters related to PMC-based axis control (2/2)

No.	Symbol	Description
0672	-----	FL feedrate for reference position return operation along a PMC controlled axis
0685	-----	F0 feedrate for independent rapid traverse override along a PMC controlled axis
0698	-----	Maximum feedrate for feed per rotation along a PMC controlled axis
7021#6	SUB1	PMC axis control on the sub-CPU side is applied to the fifth and sixth axes/the fifth axis only.

(28) Parameters related to cylindrical grinding machines (FS0/00-GCC)
(1/1)

No.	Symbol	Description
0033#0 to #3	P1S*	The skip signal used with G31 P1 is invalid/valid.
0033#4 to #7	P2S*	The skip signal used with G31 P2 is invalid/valid.
0034#0 to #3	P3S*	The skip signal used with G31 P3 is invalid/valid.
0034#4 to #7	P4S*	The skip signal used with G31 P4 is invalid/valid.
0035#0 to #3	DS*	The skip signal used with the dwell skip function is invalid/valid.
0036#0	AGLST	Slanted axis control is not exercised/is exercised.
0036#1	GSP	The G code system is type A/type B.
0036#2	ZRTM1	Movement is performed/is not performed along the Z-axis in manual reference position return operation along the X-axis.
0036#4	G98	Upon power-up, the feed-per-rotation mode/feed-per-minute mode is set.

(29) Parameters related to the PMC (1/1)

No.	Symbol	Description
0028#1	PRCMSG	The program check screen displays the remaining travel distance/message from the PMC.
0060#1	PCLDB	The baud rate used for ladder loading is 4800 [bps]/9600 [bps].
0070#6	PEXRD	The R and D areas of the PMC-M are not expanded/are expanded.
0071#6	DPCRAM	When a PMC RAM board is used, PMC LOAD MENU is displayed/is not displayed.
0356 to 0359	-----	Number of characters displayed in the remaining travel distance field on the program check screen
0476 to 0479	-----	Start PMC address where remaining travel distance field characters are set

9. PARAMETERS

For T series

(30) Parameters related to the butt-type reference position setting function
(1/1)

No.	Symbol	Description
0372 to 0375	-----	Torque limit value during execution of a butt-type reference position setting cycle
0451, 0458	-----	Butting speed used when a butt-type reference position setting cycle is executed
0459 to 0462	-----	Speed for reference position setting when a butt-type reference position setting cycle is executed
0872 to 0875	-----	Reference position setting distance used when a butt-type reference position setting cycle is executed

(31) Parameters related to the DNC (1/1)

No.	Symbol	Description
0387#0	SQDNC	In DNC operation, program restart is disabled/enabled.
0390#7	NODC3	In DNC operation, reading is performed block by block/until the buffer becomes full.
0396#0	BCC	A BCC value check is performed/is not performed.
0396#2	NCKER	An ER (RS-232-C) or TR (RS-422) check is performed/is not performed.
0396#3	ERCODE	A 4-digit hexadecimal error code is not added/is added to a negative acknowledgement.
0347	-----	Mode of connection between the CNC and host when the DNC1 interface is used
0348	-----	CNC station address when the DNC1 interface is used
0365	-----	Time-out period for the no-response timer
0366	-----	Time-out period for the EOT signal timer
0368	-----	Maximum invalid data count value
0369	-----	Maximum number of message retransmissions due to NAK
0370	-----	Maximum number of characters receivable after transmission termination processing
0490	-----	Maximum datagram (data section) length

(32) Parameters related to the M-NET (1/1)

No.	Symbol	Description
0392#2	SRL1	The M-NET character length is 7/8 bits.
0392#4	SRPE	An M-NET vertical parity check is not performed/ is performed.
0392#5	SREP	The M-NET vertical parity is odd/even.
0392#7	SRS2	The number of M-NET stop bits is 1/2.
0360	-----	M-NET DI byte length
0361	-----	M-NET DO byte length
0362	-----	M-NET station address
0363	-----	M-NET baud rate
0464	-----	Timer monitoring value after a connection preparation sequence for the local station is completed
0465	-----	Polling time monitoring value for an ordinary sequence for the local station
0466	-----	Timer monitoring value from start of SAI transmission to completion of BCC transmission
0467	-----	Timer value from completion of reception to start of transmission
0468	-----	Output destination address for the spindle orientation signal

(33) Parameters related to the Cf-axis (1/1)

No.	Symbol	Description
0031#6	ESFC	In the turning mode, a feedback pulse signal from a position detector is invalid/valid.
0031#7	CNRST	In reference position return operation, a relative coordinate along the Cf-axis is not cleared/is cleared.
0032#7	ROT10	The parameter unit for inch output is 0.1 [deg/min]/1 [deg/min].

(34) Parameters related to 2-path lathes (TT) (1/2)

No.	Symbol	Description
0046#0	RSTSW	The reset key is enabled for both tool posts/only one selected tool post.
0046#1	SBAB	On the common screen, tool post 2/tool post 1 is displayed on the upper side of the screen.
0046#2	IGNAL	When an alarm is issued on a tool post, the feed hold state is set/is not set.
0047#0	TIST	The tool post selection signal is valid/invalid.
0047#1	SP2C	Graphic drawing is common to both tool posts/ varies for each tool post.
0047#2	SEPH	Output destination of the manual handle pulse signal
0047#3	2SP	The number of spindles is 1/2.
0047#4	M3LMT	A wait M code is specified by No. 0243+100/Nos. 0243 to 999.
0047#5, #6	VR*	Start number of custom macro variables common to tool posts
0048#0, #1	TY*	Tool post coordinate system relationship
0048#2	ITO	Tool post interference check method

9. PARAMETERS

For T series

(34) Parameters related to 2-path lathes (TT) (2/2)

No.	Symbol	Description
0048#3	IFM	In the manual mode, a tool post interference check is not performed/is performed.
0048#4	IFE	A tool post interference check is performed/is not performed when satisfactory conditions are present.
0048#5	ZCLR	A tool post interference check along the Z-axis is performed/is not performed.
0049#3	WTRDY	PRDY is output with the timing specific to each tool post/simultaneously.
0075#1	STHRD	Simultaneous feed per rotation and threading on both tool posts is impossible/is possible.
0381#7	IGNSIC	A serial/analog spindle is connected to tool post 2.
0383#6	FE34	On tool post 2, Cf-axis and Y-axis control is not exercised/is exercised.
0383#7	FEPA	On tool post 2, PMC axis control is not exercised/is exercised.
0384#6	FESS	A serial spindle is not connected/is connected to tool post 2.
0384#7	FECS	On tool post 2, Cs-axis control is not exercised/is exercised.
0385#7	MEMEX	On tool post 2, the part program storage length is not expanded/is expanded.
0397#4	SPCCK	Balance cut threading with one spindle on two tool posts is disabled/enabled.
0397#5, #6	BOVR*	Balance cut override
0218	-----	Number of macro variables common to tool posts
0243	-----	Third digit (hundreds digit) of a wait M code
0249	-----	Magnification for controlling the spindle speed ratio of serial interface spindles
0257 to 0263	-----	Tool post name (first to seventh characters) displayed on the screen
0264 to 0267	-----	Axis name subscript for each axis
0598	-----	Minimum program number shared by the tool posts
0730	-----	Coordinate system shift distance between the tool posts
0768, 0769	-----	Distance between the reference point of tool post 1 and the reference point of tool post 2 for each axis

(35) Parameters related to polygon turning (1/2)

No.	Symbol	Description
0069#0	PLGN3	The third axis is not set/is set as a synchronized axis for polygon turning.
0069#1	PLGN4	The fourth axis is not set/is set as a synchronized axis for polygon turning.
0069#2	PLG7	The seventh axis is not set/is set as a synchronized axis for polygon turning.
0069#3	PLG8	The eighth axis is not set/is set as a synchronized axis for polygon turning.

(35) Parameters related to polygon turning (2/2)

No.	Symbol	Description
0069#7	PLHZ	Reference position return operation along a synchronized axis with G28 is performed by rapid traverse/always at low speed.
0667	-----	Maximum allowable speed of a synchronized axis
0668	-----	Speed of a synchronized axis
0669	-----	Jog feedrate when the rotary switch for a synchronized axis is positioned at 10
0670	-----	F0 feedrate for rapid traverse override of a synchronized axis
0671	-----	FL feedrate for reference position return along a synchronized axis
0778	-----	Travel distance per synchronized axis rotation

(36) Parameters related to synchronized control/axis recomposition (1/1)

No.	Symbol	Description
0380#0 to #5	SMR※	Synchronized control with mirror image on is not exercised/is exercised.
0380#6	SPERR	An acceleration/deceleration difference is not added/is added to an out-of-sync distance in synchronized axis recomposition.
0380#7	NRST	Synchronized axis recomposition is canceled/is not canceled upon reset.
0381#0 to #5	SER※	If an excessive position deviation occurs in synchronized axis recomposition, an alarm is not issued/is issued.
0382#0 to #5	PKU※	When parking is performed during synchronized axis recomposition, absolute and relative coordinates are not updated/are updated.
0383#0 to #5	MPM※	When synchronized axis recomposition is started, a workpiece coordinate system is not set/is set automatically.
0384#0 to #5	MPS※	At the end of synchronized axis recomposition, the workpiece coordinate system is not reset/is reset to the original one.
0385#0 to #5	MCD※	Axis recomposition with mirror image on is not performed/is performed.
0320 to 0323	-----	Master axis with which each axis is synchronized
0326 to 0329	-----	Axis of tool post 1 for which the axis of tool post 2 is used to perform axis recomposition
0686 to 0691	-----	Limit on position deviation difference in synchronized control where each axis serves as a slave axis
0692 to 0697	-----	Position deviation difference in synchronized control where each axis serves as a slave axis
0860 to 0865	-----	Reference position coordinates in the coordinate system in axis recomposition for each axis

10. ALARM LIST

For M series

(1) Program errors (P/S alarm)

(a) M series

Number	Message	Contents
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
003	TOO MANY DIGITS	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	ADDRESS NOT FOUND	A numeral or the sign "—" was input without an address at the beginning of a block. Modify the program.
005	NO DATA AFTER ADDRESS	The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.
006	ILLEGAL USE OF NEGATIVE SIGN	Sign "—" input error (Sign "—" was input after an address with which it cannot be used. Or two or more "—" signs were input.) Modify the program.
007	ILLEGAL USE OF DECIMAL POINT	Decimal point "." input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.
009	ILLEGAL ADDRESS INPUT	Unusable character was input in significant area. Modify the program.
010	IMPROPER G-CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.
014	ILLEGAL LEAD COMMAND	In variable lead threading, the lead incremental and decremental outputted by address K exceeded the maximum command value or a command such that the lead becomes a negative value is given. Modify the program.
015	TOO MANY AXES COMMANDED	An attempt was made to move the machine along the axes, but the number of the axes exceeded the specified number of axes controlled simultaneously. Alternatively, in a block where the skip function activated by the torque-limit reached signal (G31 P99/P98) was specified, either moving the machine along an axis was not specified, or moving the machine along multiple axes was specified. Specify movement only along one axis.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 3410.
021	ILLEGAL PLANE AXIS COMMANDED	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.

Number	Message	Contents
023	ILLEGAL RADIUS COMMAND	In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.
029	ILLEGAL OFFSET VALUE	The offset values specified by T code is too large. Modify the program.
030	ILLEGAL OFFSET NUMBER	The offset number in T function specified for tool offset is tool large. Modify the program.
031	ILLEGAL P COMMAND IN G10	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. Modify the program.
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
033	NO SOLUTION AT CRC	A point of intersection cannot be determined for tool nose radius compensation. Modify the program.
034	NO CIRC ALLOWED IN ST-UP /EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation. Modify the program.
035	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN NRC	The offset plane is switched in tool nose radius compensation. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
039	CHF/CNR NOT ALLOWED IN NRC	Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. Modify the program.
040	INTERFERENCE IN G90/G94 BLOCK	Overcutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program.
041	INTERFERENCE IN NRC	Overcutting will occur in tool nose radius compensation. Modify the program.
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command.
050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
051	MISSING MOVE AFTER CHF/CNR	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
052	CODE IS NOT G01 AFTER CHF/CNR	The block next to the chamfering or corner R block is not G01. Modify the program.

10. ALARM LIST

For M series

Number	Message	Contents
053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma(",") is not C or R in direct drawing dimensions programming. Modify the program.
054	NO TAPER ALLOWED AFTER CHF/CNR	A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.
055	MISSING MOVE VALUE IN CHF/CNR	In chamfering or corner R block, the move distance is less than chamfer or corner R amount.
056	NO END POINT & ANGLE IN CHF/CNR	Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering command, I(K) is commanded for the X(Z) axis.
057	NO SOLUTION OF BLOCK END	Block end point is not calculated correctly in direct dimension drawing programming.
058	END POINT NOT FOUND	Block end point is not found in direct dimension drawing programming.
059	PROGRAM NUMBER NOT FOUND	In an external program number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background editing.
060	SEQUENCE NUMBER NOT FOUND	Commanded sequence number was not found in the sequence number search. Check the sequence number.
061	ADDRESS P/Q NOT FOUND IN G70-G73	Address P or Q is not specified in G70, G71, G72, or G73 command. Modify the program.
062	ILLEGAL COMMAND IN G71-G76	<ol style="list-style-type: none"> 1 The depth of cut in G71 or G72 is zero or negative value. 2 The repetitive count in G73 is zero or negative value. 3 the negative value is specified to Δi or Δk is zero in G74 or G75. 4 A value other than zero is specified to address U or W, though Δi or Δk is zero in G74 or G75. 5 A negative value is specified to Δd, though the relief direction in G74 or G75 is determined. 6. Zero or a negative value is specified to the height of thread or depth of cut of first time in G76. 7 The specified minimum depth of cut in G76 is greater than the height of thread. 8 An unusable angle of tool tip is specified in G76. Modify the program.
063	SEQUENCE NUMBER NOT FOUND	The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. Modify the program.
064	SHAPE PROGRAM NOT MONOTONOUSLY	A target shape which cannot be made by monotonic machining was specified in a repetitive canned cycle (G71 or G72).

Number	Message	Contents
065	ILLEGAL COMMAND IN G71–G73	<p>1 G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command.</p> <p>2. Address Z(W) or X(U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively.</p> <p>Modify the program.</p>
066	IMPROPER G–CODE IN G71–G73	An unallowable G code was commanded between two blocks specified by address P in G71, G72, or G73. Modify the program.
067	CAN NOT ERROR IN MDI MODE	G70, G71, G72, or G73 command with address P and Q. Modify the program.
068	MORE THAN 10 POCKETS HAVE BEEN SPECIFIED.	More than 10 pockets have been specified as type II in a G71 or G72 code.
069	FORMAT ERROR IN G70–G73	the final move command in the blocks specified by P and Q of G70, G71, G72, and G73 ended with chamfering or corner R. Modify the program.
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
071	DATA NOT FOUND	The address to be searched was not found. Or the program with specified program number was not found in program number search. Check the data.
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option), 200 (option), or 400 (option). Delete unnecessary programs and execute program registration again.
073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registration again.
074	ILLEGAL PROGRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
075	PROTECT	An attempt was made to register a program whose number was protected.
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	The subprogram was called in five folds. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. Otherwise, a called program is being edited in background processing. Correct the program, or discontinue the background editing.
079	PROGRAM VERIFY ERROR	In memory or program collation, a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device.

10. ALARM LIST

For M series

Number	Message	Contents
080	G37 ARRIVAL SIGNAL NOT ASSERTED	In the automatic tool compensation function (G36, G37), the measurement position reach signal (XAE or ZAE) is not turned on within an area specified in parameter 6254 (value ϵ). This is due to a setting or operator error.
081	OFFSET NUMBER NOT FOUND IN G37	Automatic tool compensation (G36, G37) was specified without a T code. (Automatic tool compensation function) Modify the program.
082	T-CODE NOT ALLOWED IN G37	T code and automatic tool compensation (G36, G37) were specified in the same block. (Automatic tool compensation function) Modify the program.
083	ILLEGAL AXIS COMMAND IN G37	In automatic tool compensation (G36, G37), an invalid axis was specified or the command is incremental. Modify the program.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
090	REFERENCE RETURN INCOMPLETE	The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. Check the program contents.
091	MANUAL RETURN IMPOSSIBLE DURING PAUSE	A manual return to the reference position was impossible because of the program being at pause. Press the reset button to cause a manual return.
092	AXES NOT ON THE REFERENCE POINT	The commanded axis by G27 (Reference position return check) did not return to the reference position.
094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the operator's manual.
095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.) Perform the correct operation according to the operator's manual.
096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.) Perform the correct operation according to the operator's manual.

Number	Message	Contents
097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P / S 94 to 97 reset, no automatic operation is performed.) Perform automatic operation.
098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
100	PARAMETER WRITE ENABLE	On the PARAMETER(SETTING) screen, PWE(parameter writing enabled) is set to 1. Set it to 0, then reset the system.
101	PLEASE CLEAR MEMORY	The power turned off while rewriting the memory by program edit operation. If this alarm has occurred, press <RESET> while pressing <PROG>, and only the program being edited will be deleted. Register the deleted program.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.
111	CALCULATED DATA OVERFLOW	The result of calculation turns out to be invalid, an alarm No.111 is issued. -10 ⁴⁷ to -10 ⁻²⁹ , 0, 10 ⁻²⁹ to 10 ⁴⁷ Modify the program.
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°) Modify the program.
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	There is an error in other formats than <Formula>. Modify the program.
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro or in high-speed cycle machining. The header contents are improper. This alarm is given in the following cases: High speed cycle machining <ol style="list-style-type: none"> 1. The header corresponding to the specified machining cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 – 999). 3. The number of data in the header is out of the allowable range (0 – 32767). 4. The start data variable number of executable format data is out of the allowable range (#20000 – #85535). 5. The last storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data is overlapped with the variable number used in the header. Modify the program.

10. ALARM LIST

For M series

Number	Message	Contents
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.
119	ILLEGAL ARGUMENT	The SQRT argument is negative. Or BCD argument is negative, and other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	DUPLICATE MACRO MODAL-CALL	The macro modal call is specified in double. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	<Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, $1 \leq n \leq 3$ is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation> is used. Modify the program.
130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.
135	SPINDLE ORIENTATION PLEASE	Without any spindle orientation, an attempt was made for spindle indexing. Perform spindle orientation.
136	C/H-CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as spindle indexing addresses C, H. Modify the program.
137	M-CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as M-code related to spindle indexing. Modify the program.
139	CAN NOT CHANGE PMC CONTROL AXIS	An axis is selected in commanding by PMC axis control. Modify the program.

Number	Message	Contents
145	ILLEGAL CONDITIONS IN POLAR COORDINATE INTERPOLATION	The conditions are incorrect when the polar coordinate interpolation starts or it is canceled. 1) In modes other than G40, G12.1/G13.1 was specified. 2) An error is found in the plane selection. Parameters No. 5460 and No. 5461 are incorrectly specified. Modify the value of program or parameter.
146	IMPROPER G CODE	G codes which cannot be specified in the polar coordinate interpolation mode was specified. Modify the program.
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. exceeds the maximum allowable value. Modify the program.
151	TOOL GROUP NUMBER NOT FOUND	The tool group commanded in the machining program is not set. Modify the value of program or parameter.
152	NO SPACE FOR TOOL ENTRY	The number of tools within one group exceeds the maximum value registerable. Modify the number of tools.
153	T-CODE NOT FOUND	In tool life data registration, a T code was not specified where one should be. Correct the program.
155	ILLEGAL T-CODE IN M06	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
156	P/L COMMAND NOT FOUND	P and L commands are missing at the head of program in which the tool group is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups to be set exceeds the maximum allowable value. Modify the program.
158	ILLEGAL TOOL LIFE DATA	The tool life to be set is too excessive. Modify the setting value.
159	TOOL DATA SETTING INCOMPLETE	During executing a life data setting program, power was turned off. Set again.
160	MISMATCH WAITING M-CODE (TT only)	Different M code is commanded in heads 1 and 2 as waiting M code. Modify the program.
163	COMMAND G68/G69 INDEPENDENTLY (TT only)	G68 and G69 are not independently commanded in balance cut. Modify the program.
169	ILLEGAL TOOL GEOMETRY DATA (TT only)	Incorrect tool figure data in interference check.
175	ILLEGAL G107 COMMAND	Conditions when performing circular interpolation start or cancel not correct. Modify the program.

10. ALARM LIST

For M series

Number	Message	Contents
176	IMPROPER G-CODE IN G107	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning: G28, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G50, G52 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.
177	CHECK SUM ERROR (G05 MODE)	Check sum error Modify the program.
178	G05 COMMAN- DED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAM. (NO. 7510) SETTING ERROR	The number of controlled axes set by the parameter (No. 597) exceeds the maximum number. Modify the parameter setting value.
180	COMMUNICATION ERROR (REMOTE BUF)	Remote buffer connection alarm has generated. Confirm the number of cables, parameters and I/O device.
194	SPINDLE COMMAND IN SYNCHRO-MODE	A contour control mode, spindle positioning (Cs-axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.
195	SPINDLE CONTROL MODE SWITCH	The serial spindle control mode was not switched. Check the PMC ladder program.
197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the C-axis when the signal CON(DGN=G027#7) was off. Correct the program, or consult the PMC ladder diagram to find the reason the signal is not turned on.
199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value is out of the range or is not specified. The range for S values which can be specified in rigid tapping is set in parameter 5243. Change the setting in the parameter or modify the program.
201	FEEDRATE NOT FOUND IN RIGID TAP	In the rigid tap, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large.
203	PROGRAM MISS AT RIGID TAPPING	In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 (G74) block. Modify the program.
205	RIGID MODE DI SIGNAL OFF	Rigid mode DI signal is not ON when G84 (G74) is executed though the rigid M code (M29) is specified. Consult the PMC ladder diagram to find the reason the DI signal (DGNG061.1) is not turned on.

Number	Message	Contents
210	CAN NOT COMMAND M198/M199	M198 and M199 are executed in the schedule operation. M198 is executed in the DNC operation. Modify the program.
211	G31 (HIGH) NOT ALLOWED IN G99	G31 is commanded in the per revolution command when the high-speed skip option is provided. Modify the program.
212	ILLEGAL PLANE SELECT	The direct drawing dimensions programming is commanded for the plane other than the Z-X plane. Correct the program.
213	ILLEGAL COMMAND IN SYNCHRO-MODE	Movement is commanded for the axis to be synchronously controlled.
214	ILLEGAL COMMAND IN SYNCHRO-MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
217	DUPLICATE G251 (COMMANDS)	G251 is further commanded in the G251 mode. Modify the program.
218	NOT FOUND P/Q COMMAND IN G251	P or Q is not commanded in the G251 block, or the command value is out of the range. Modify the program.
219	COMMAND G250/G251 INDEPENDENTLY	G251 and G250 are not independent blocks.
220	ILLEGAL COMMAND IN SYNCHRO-MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis.
221	ILLEGAL COMMAND IN SYNCHRO-MODE	Polygon machining synchronous operation and axis control or balance cutting are executed at a time. Modify the program.
224	RETURN TO REFERENCE POINT	Not returned to reference point before cycle start.
225	SYNCHRONOUS/MIXED CONTROL ERROR (TT only)	This alarm is generated in the following circumstances. (Searched for during synchronous and mixed control command. 1 When there is a mistake in axis number parameter setting. 2 When there is a mistake in control commanded. Modify the program or the parameter.
226	ILLEGAL COMMAND IN SYNCHRO-MODE (TT only)	A travel command has been sent to the axis being synchronized in synchronous mode. Modify the program or the parameter.
229	CAN NOT KEEP SYNCHRO-STATE (TT only)	This alarm is generated in the following circumstances. 1 When the synchro/mixed state could not be kept due to system overload. 2 The above condition occurred in CMC devices (hardware) and synchro-state could not be kept. (This alarm is not generated in normal use conditions.)
233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS-232-C interface, other users were using it.

10. ALARM LIST

For T series

(b) T series

Number	Meaning	Contents and remedy
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
003	TOO MANY DIGITS	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	ADDRESS NOT FOUND	A numeral or the sign " - " was input without an address at the beginning of a block. Modify the program .
005	NO DATA AFTER ADDRESS	The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.
006	ILLEGAL USE OF NEGATIVE SIGN	Sign " . " input error (Sign " - " was input after an address with which it cannot be used. Or two or more " - " signs were input.) Modify the program.
007	ILLEGAL USE OF DECIMAL POINT	Decimal point " - " input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.
008	ILLEGAL USE OF PROGRAM END	An attempt was made to execute EOR (%) because there was not M02, M30, or M99 at the end of the program. Correct the program.
009	ILLEGAL ADDRESS INPUT	Unusable character was input in significant area. Modify the program.
010	IMPROPER G-CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.
014	CAN NOT COMMAND G95	A synchronous feed is specified without the option for threading / synchronous feed.
015	TOO MANY AXES COMMANDED	The number of the commanded axes exceeded that of simultaneously controlled axes.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 876.
021	ILLEGAL PLANE AXIS COMMANDED	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.
025	CANNOT COMMAND F0 IN G02/G03	F0 (fast feed) was instructed by F1 -digit column feed in circular interpolation. Modify the program.
027	NO AXES COMMANDED IN G43/G44	No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Modify the program.

Number	Meaning	Contents and remedy
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.
029	ILLEGAL OFFSET VALUE	The offset values specified by H code is too large. Modify the program.
030	ILLEGAL OFFSET NUMBER	The offset number specified by D/H code for tool length offset or cutter compensation is too large. Modify the program.
031	ILLEGAL P COMMAND IN G10	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. Modify the program.
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
033	NO SOLUTION AT CRC	A point of intersection cannot be determined for cutter compensation C. Modify the program.
034	NO CIRC ALLOWED IN ST-UP /EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation. Modify the program.
035	CAN NOT COMMANDED G39	G39 is commanded in cutter compensation B cancel mode or on the plane other than offset plane. Modify the program.
036	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN CRC	G40 is commanded on the plane other than offset plane in cutter compensation B. The plane selected by using G17, G18 or G19 is changed in cutter compensation C mode. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center. Modify the program.
041	INTERFERENCE IN CRC	Overcutting will occur in cutter compensation C. Two or more blocks are consecutively specified in which functions such as the auxiliary function and dwell functions are performed without movement in the cutter compensation mode. Modify the program.
042	G45/G48 NOT ALLOWED IN CRC	Tool offset (G45 to G48) is commanded in cutter compensation. Modify the program.
043	ILLEGAL T-CODE COMMAND	In the DRILL-MATE, a T code was not specified together with the M06 code in a block. Alternatively, the Tcode was out of range.
044	G27-G30 NOT ALLOWED IN FIXED CYC	One of G27 to G30 is commanded in canned cycle mode. Modify the program.
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command.
050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
051	MISSING MOVE AFTER CHF/CNR	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.

10. ALARM LIST

For T series

Number	Meaning	Contents and remedy
052	CODE IS NOT G01 AFTER CHF/CNR	The block next to the chamfering or corner R block is not G01. Modify the program.
053	TOO MANY ADDRESS COMMANDS	For systems without the arbitrary angle chamfering or corner R cutting, a comma was specified. For systems with this feature, a comma was followed by something other than R or C. Correct the program.
055	MISSING MOVE VALUE IN CHF/ CNR	In the arbitrary angle chamfering or corner R block, the move distance is less than chamfer or corner R amount.
058	END POINT NOT FOUND	In a arbitrary angle chamfering or corner R cutting block, a specified axis is not in the selected plane. Correct the program.
059	PROGRAM NUMBER NOT FOUND	In an external program number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background editing.
060	SEQUENCE NUMBER NOT FOUND	Commanded sequence number was not found in the sequence number search. Check the sequence number.
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
071	DATA NOT FOUND	The address to be searched was not found. Or the program with specified program number was not found in program number search. Check the data.
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option) or, 200 (option). Delete unnecessary programs and execute program registration again.
073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registration again.
074	ILLEGAL PRO- GRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	A level larger than the maximum allowable nesting level of subprogram calls was specified. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. An attempt was made to call a program being edited in the background processing mode. Correct the program.
079	PROGRAM VERIFY ERROR	In memory or program collation, a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device.

Number	Meaning	Contents and remedy
080	G37 ARRIVAL SIGNAL NOT ASSERTED	In the automatic tool length measurement function (G37), the measurement position reach signal (XAE, YAE, or ZAE) is not turned on within an area specified in parameter (value ε). This is due to a setting or operator error.
081	OFFSET NUMBER NOT FOUND IN G37	Tool length automatic measurement (G37) was specified without a H code. (Automatic tool length measurement function) Modify the program.
082	H-CODE NOT ALLOWED IN G37	H code and automatic tool compensation (G37) were specified in the same block. (Automatic tool length measurement function) Modify the program.
083	ILLEGAL AXIS COMMAND IN G37	In automatic tool length measurement, an invalid axis was specified or the command is incremental. Modify the program.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
090	REFERENCE RETURN INCOMPLETE	The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return.
091	MANUAL RETURN IMPOSSIBLE DURING PAUSE	A manual return to the reference position was impossible because of the program being at pause. Press the reset button to cause a manual return.
092	AXES NOT ON THE REFERENCE POINT	The commanded axis by G28 (automatic reference position return) or G27 (reference position return check) did not return to the reference position.
094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the operator's manual.
095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.)
096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.)

10. ALARM LIST

For T series

Number	Meaning	Contents and remedy
097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P / S 94 to 97 reset, no automatic operation is performed.) Perform automatic operation.
098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search.
099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
100	PARAMETER WRITE ENABLE	On the PARAMETER(SETTING) screen, PWE (parameter writing enabled) is set to 1. Set it to 0, then reset the system.
101	PLEASE CLEAR MEMORY	The power turned off while rewriting the memory by program edit operation. When this alarm occurs, set the PWE parameter to 1, then switch on the power while holding down the <DELET> key. All programs will be deleted.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.
111	CALCULATED DATA OVERFLOW	The result of calculation turns out to be invalid, an alarm No.111 is issued. -10 ⁴⁷ to -10 ⁻²⁹ , 0, 10 ⁻²⁹ to 10 ⁴⁷
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°)
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	Custom macro A specified an undefined H code in a G65 block. There is an error in other formats than <Formula>. Modify the program.
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro or in high-speed cycle machining. The header contents are improper. This alarm is given in the following cases: High speed cycle machining 1. The header corresponding to the specified machining cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 - 999). 3. The number of data in the header is out of the allowable range (0 - 32767). 4. The start data variable number of executable format data is out of the allowable range (#20000 - #85535). 5. The last storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data is overlapped with the variable number used in the header. Modify the program.
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.

Number	Meaning	Contents and remedy
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.
119	ILLEGAL ARGUMENT	The SQRT argument is negative. Or BCD argument is negative, and other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	DUPLICATE MACRO MODAL-CALL	The macro modal call is specified in double. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	Custom macro A specified an undefined H code in a G65 block. <Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DO n, $1 \leq n \leq 3$ is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation> is used. Modify the program.
130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.
135	ILLEGAL ANGLE COMMAND	The index table indexing positioning angle was instructed in other than an integral multiple of the value of the minimum angle. Modify the program.
136	ILLEGAL AXIS COMMAND	In index table indexing. Another control axis was instructed together with the B axis. Modify the program.
139	CAN NOT CHANGE PMC CONTROL AXIS	An axis is selected in commanding by PMC axis control. Modify the program.
141	CAN NOT COMMAND G51 IN CRC	G51 (Scaling ON) is commanded in the tool off-set mode. Modify the program.
142	ILLEGAL SCALE RATE	Scaling magnification is commanded in other than 1 – 999999. Correct the scaling magnification setting .

10. ALARM LIST

For T series

Number	Meaning	Contents and remedy
143	SCALED MOTION DATA OVERFLOW	The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. Correct the program or scaling magnification.
144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.
148	ILLEGAL SETTING DATA	Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters (No.1710 to No.1714)
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. exceeds the maximum allowable value in the tool life management. Modify the program.
151	TOOL GROUP NUMBER NOT FOUND	The tool group of the tool life management commanded in the machining program is not set. Modify the value of program or parameter.
152	NO SPACE FOR TOOL ENTRY	The number of tools within one group in the tool life management exceeds the maximum value registerable. Modify the number of tools.
153	T-CODE NOT FOUND	In tool life data registration, a T code was not specified where one should be. Correct the program.
154	NOT USING TOOL IN LIFE GROUP	When the group is not commanded in the tool life management, H99 or D99 was commanded. Correct the program.
155	ILLEGAL T-CODE IN M06	In the machining program, M06 and T code in the same block do not correspond to the tool life management group number in use. Correct the program.
156	P/L COMMAND NOT FOUND	P and L commands are missing at the head of program in which the tool group of the tool life management is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups in the tool life management to be set exceeds the maximum allowable value. Modify the program.
158	ILLEGAL TOOL LIFE DATA	The tool life to be set is too excessive. Modify the setting value.
159	TOOL DATA SETTING INCOMPLETE	During executing a life data setting program in tool life management, power was turned off. Set again.
175	ILLEGAL G107 COMMAND	Conditions when performing circular interpolation start or cancel not correct. To change the mode to the cylindrical interpolation mode, specify the command in a format of "G07.1 rotation-axis name radius of cylinder."
176	IMPROPER G-CODE IN G107	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning: G28,, G73, G74, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G52,G92, 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.
177	CHECK SUM ERROR (G05 MODE)	Check sum error is occurred in the high-speed remote buffer.

Number	Meaning	Contents and remedy
178	G05 COMMAND IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAMETER SETTING ERROR	The number of controlled axes set by the parameter 7510 exceeds the maximum number. Modify the parameter setting value.
180	COMMUNICATION ERROR (REMOTE BUF)	Remote buffer connection alarm has generated. Confirm the number of cables, parameters and I/O device.
181	FORMAT ERROR IN G81 BLOCK	G81 block format error (hobbing machine) 1) T (number of teeth) has not been instructed. 2) Data outside the command range was instructed by either T, L, Q or P. Modify the program.
182	G81 NOT COMMANDED	G83 (C axis servo lag quantity offset) was instructed though synchronization by G81 has not been instructed. Correct the program. (hobbing machine)
183	DUPLICATE G83 (COMMANDS)	G83 was instructed before canceled by G82 after compensating for the C axis servo lag quantity by G83. (hobbing machine)
184	ILLEGAL COMMAND IN G81	A command not to be instructed during synchronization by G81 was instructed. (hobbing machine) 1) A C axis command by G00, G27, G28, G29, G30, etc. was instructed. 2) Inch/Metric switching by G20, G21 was instructed.
185	RETURN TO REFERENCE POINT	G81 was instructed without performing reference position return after power on or emergency stop. (hobbing machine) Perform reference position return.
186	PARAMETER SETTING ERROR	Parameter error regarding G81 (hobbing machine) 1) The C axis has not been set to be a rotary axis. 2) A hob axis and position coder gear ratio setting error
190	ILLEGAL AXIS SELECT	In the constant surface speed control, the specified axis command (P) contains an illegal value. Correct the program.
194	SPINDLE COMMAND IN SYNCHRO-MODE	Cs contour control or rigid tapping was specified during serial spindle synchronous control. Correct the program.
195	SPINDLE CONTROL MODE SWITCH	The serial spindle control mode was not switched. Check the PMC ladder program.
197	C-AXIS COMMANDED IN SPINDLE MODE	A command for Cs-axis movement was issued when the current control mode is not serial spindle Cs contour control. Check the PMC ladder program or machining program.
199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value is out of the range or is not specified. Modify the program.

10. ALARM LIST

For T series

Number	Meaning	Contents and remedy
201	FEEDRATE NOT FOUND IN RIGID TAP	In the rigid tap, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large.
203	PROGRAM MISS AT RIGID TAPPING	In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 (G74) block. Modify the program.
205	RIGID MODE DI SIGNAL OFF	Rigid mode DI signal is not ON when G84 (G74) is executed though the rigid M code (M29) is specified. Consult the PMC ladder diagram to find the reason the DI signal (DGNG061.1) is not turned on. Modify the program.
206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Correct the program.
210	CAN NOT COMMAND M198/M199	M198 and M199 are executed in the schedule operation. M198 is executed in the DNC operation.
211	CAN NOT COMMAND HIGH-SPEED SKIP	A high-speed skip (G31) was specified during the feed-per-rotation or rigid tapping mode. Correct the program.
212	ILLEGAL PLANE SELECT	The arbitrary angle chamfering or a corner R is commanded or the plane including an additional axis. Correct the program.
213	ILLEGAL COMMAND IN SYNCHRO-MODE	Any of the following alarms occurred in the operation with the simple synchronization control. 1) The program issued the move command to the slave axis. 2) The program issued the manual continuous feed/manual handle feed/incremental feed command to the slave axis. 3) The program issued the automatic reference position return command without executing the manual reference position return after the power was turned on. 4) The difference between the position error amount of the master and slave axes exceeded the value specified in parameter .
214	ILLEGAL COMMAND IN SYNCHRO-MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
222	DNC OP. NOT ALLOWED IN BG.-EDIT	Input and output are executed at a time in the background edition. Execute a correct operation.
224	RETURN TO REFERENCE POINT	Reference position return has not been performed before the automatic operation starts. Perform reference position return.
230	R CODE NOT FOUND	The infeed quantity R has not been instructed for the G160 block of the canned grinding cycle. Or the R command value is negative. Correct the program.
250	SIMULTANEOUS M06 AND Z-AXIS MOVEMENT NOT ALLOWED	A tool change (M06) and a Z-axis movement were specified simultaneously in the DRILL MATE. Correct the program.

(2) Background edit alarm

Number	Meaning	Contents and remedy
???	BACKGROUND EDIT ALARM	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit. (070, 071, 072, 073, 074 085,086,087 etc.)
140	SELECTED PROGRAM ALARM	It was attempted to select or delete in the background a program being selected in the foreground. (Note) Use background editing correctly.

NOTE A background editing alarm may occur even during MDI operation-B. This is because MDI operation-B uses the background editing function.

(3) Absolute pulse coder (APC) alarm

Number	Meaning	Contents and remedy
3n0	nth-axis origin return	Manual reference position return is required for the nth-axis.
3n1	APC alarm: nth-axis communication	nth-axis APC communication error. Failure in data transmission Possible causes include a faulty APC, cable, or servo interface module.
3n2	APC alarm: nth-axis over time	nth-axis APC overtime error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
3n3	APC alarm: nth-axis framing	nth-axis APC framing error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
3n4	APC alarm: nth-axis parity	nth-axis APC parity error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
3n5	APC alarm: nth-axis pulse error	nth-axis APC pulse error alarm. APC alarm.APC or cable may be faulty.
3n6	APC alarm: nth-axis battery voltage 0	nth-axis APC battery voltage has decreased to a low level so that the data cannot be held. APC alarm. Battery or cable may be faulty.
3n7	APC alarm: nth-axis battery low 1	nth-axis axis APC battery voltage reaches a level where the battery must be renewed. APC alarm. Replace the battery.
3n8	APC alarm: nth-axis battery low 2	nth-axis APC battery voltage has reached a level where the battery must be renewed (including when power is OFF). APC alarm.

10. ALARM LIST

(4) Serial pulse coder (SPC) alarms

When either of the following alarms is issued, a possible cause is a faulty serial pulse coder or cable.

Number	Meaning	Contents
3n9	SPC ALARM: n AXIS PULSE COD- ER	The n axis (axis 1–8) pulse coder has a fault.

• The details of serial pulse coder alarm No.3n9

The details of serial pulse coder alarm No. 3n9 are displayed in the diagnosis display (No. 760 to 767, 770 to 777) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
760 to 767		CSAL	BLAL	PHAL	RCAL	BZAL	CKAL	SPHL

CSAL : The serial pulse coder is defective. Replace it.

BLAL : The battery voltage is low. Replace the batteries. This alarm has nothing to do with alarm (serial pulse coder alarm).

SPHL : The serial pulse coder or feedback cable is defective. Replace the serial pulse coder or cable.

RCAL : The serial pulse coder is defective. Replace it.

BZAL : The pulse coder was supplied with power for the first time. Make sure that the batteries are connected.

Turn the power off, then turn it on again and perform a reference position return. This alarm has nothing to do with alarm (serial pulse coder alarm).

CKAL : The serial pulse coder is defective. Replace it.

PHAL : The serial pulse coder or feedback cable is defective. replace the serial pulse coder or cable.

	#7	#6	#5	#4	#3	#2	#1	#0
770 to 777	DTERR	CRCERR	STBERR					

DTERR : The serial pulse coder encountered a communication error. The pulse coder, feedback cable, or feedback receiver circuit is defective. Replace the pulse coder, feedback cable, or NC-axis board

CRCERR : The serial pulse coder encountered a communication error. The pulse coder, feedback cable, or feedback receiver circuit is defective. Replace the pulse coder, feedback cable, or NC-axis board.

STBERR : the serial pulse coder encountered a communication error. The pulse coder, feedback cable, or feedback receiver circuit is defective. Replace the pulse coder, feedback cable, or NC-axis board.

(5) Servo alarms

Number	Meaning	Contents and actions
400	SERVO ALARM: 1, 2TH AXIS OVERLOAD	1-axis, 2-axis overload signal is on. Refer to diagnosis display No. 720 or 721 for details.
401	SERVO ALARM: 1, 2TH AXIS VRDY OFF	1-axis, 2-axis servo amplifier READY signal (DRDY) went off.
402	SERVO ALARM: 3, 4TH AXIS OVERLOAD	3-axis, 4-axis overload signal is on. Refer to diagnosis display No. 722 or 723 for details.
403	SERVO ALARM: 3, 4TH AXIS VRDY OFF	3-axis, 4-axis servo amplifier READY signal (DRDY) went off.
404	SERVO ALARM: n-TH AXIS VRDY ON	Even though the n-th axis (axis 1-8) READY signal (MCON) went off, the servo amplifier READY signal (DRDY) is still on. Or, when the power was turned on, DRDY went on even though MCON was off. Check that the axis card and servo amplifier are connected.
405	SERVO ALARM: ZERO POINT RETURN FAULT	Position control system fault. Due to an NC or servo system fault in the reference position return, there is the possibility that reference position return could not be executed correctly. Try again from the manual reference position return.
406	SERVO ALARM: 7, 8TH AXIS OVERLOAD 7, 8TH AXIS VRDY OFF	7-axis, 8-axis overload signal is on. Refer to diagnosis display No. 726 or 727 for details. 3-axis, 4-axis servo amplifier READY signal (DRDY) went off.
4n0	SERVO ALARM: n-TH AXIS - EXCESS ERROR	The position deviation value when the n-th axis stops is larger than the set value. Note) Limit value must be set to parameter for each axis.
4n1	SERVO ALARM: n-TH AXIS - EXCESS ERROR	The position deviation value when the n-th axis moves is larger than the set value. Note) Limit value must be set to parameter for each axis.
4n3	SERVO ALARM: n-TH AXIS - LSI OVERFLOW	The contents of the error register for the n-th axis exceeded $\pm 2^{31}$ power. This error usually occurs as the result of an improperly set parameters.
4n4	SERVO ALARM: n-TH AXIS - DETECTION RELATED ERROR	N-th axis digital servo system fault. Refer to diagnosis display No. 720 and No.727 for details.
4n5	SERVO ALARM: n-TH AXIS - EXCESS SHIFT	A speed higher than 4000000 units/s was attempted to be set in the n-th axis. This error occurs as the result of improperly set CMR.
4n6	SERVO ALARM: n-TH AXIS - DIS-CONNECTION	Position detection system fault in the n-th axis pulse coder (disconnection alarm).

10. ALARM LIST

Number	Meaning	Contents and actions
4n7	SERVO ALARM: n-TH AXIS - PARAMETER INCORRECT	<p>This alarm occurs when the n-th axis is in one of the conditions listed below. (Digital servo system alarm)</p> <ol style="list-style-type: none"> 1) The value set in Parameter No. 8n20 (motor form) is out of the specified limit. 2) A proper value (111 or -111) is not set in parameter No. 8n22 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 8n23 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 8n24 (number of position feedback pulses per motor revolution). 5) Parameters No. 8n84 and No. 8n85 (flexible field gear rate) have not been set. 6) An axis selection parameter (from No. 269 to 274) is incorrect. 7) An overflow occurred during parameter computation.
490	SERVO ALARM: 5TH AXIS OVERLOAD	5-axis, 6-axis overload signal is on. Refer to diagnosis display No. 724 or 725 for details.
491	SERVO ALARM: 5, 6TH VRDY OFF	5-axis, 6-axis servo amplifier READY signal (DRDY) went off.
494	SERVO ALARM: 5, 6TH AXIS VRDY ON	The axis card ready signal (MCON) for axes 5 and 6 is off, but the servo amplifier ready signal (DRDY) is not. Alternatively, when the power is applied, the DRDY is on, but the MCON is not. Ensure that the axis card and servo amplifier are connected.
495	SERVO ALARM: 5, 6TH AXIS ZERO POINT RETURN	This is a position control circuit error. It is likely that a return to the reference position failed because of an error in the NC or the servo system. Retry a return to the reference position.

NOTE If an excessive spindle error alarm occurs during rigid tapping, the relevant alarm number for the tapping feed axis is displayed.

• Details of servo alarm No.4n4

The detailed descriptions of servo alarm number 414 are displayed with diagnosis numbers 720 to 727 in the sequence of axis numbers.

	#7	#6	#5	#4	#3	#2	#1	#0
720 to 727	OVL	LV	OVC	HCAL	HVA	DCAL	FBAL	OFAL

OVL : An overload alarm is being generated.

(This bit causes servo alarm No. 400, 402, 406, 490).

LV : A low voltage alarm is being generated in servo amp.
Check LED.

OVC : A overcurrent alarm is being generated inside of digital servo.

HCAL : An abnormal current alarm is being generated in servo amp.
Check LED.

HVAL : An overvoltage alarm is being generated in servo amp.
Check LED.

DCAL : A regenerative discharge circuit alarm is being generated in servo amp.
Check LED.

FBAL : A disconnection alarm is being generated.
(This bit causes servo alarm No.4n6.)

OFAL : An overflow alarm is being generated inside of digital servo.

(6) Spindle alarms

Number	Meaning	Contents and remedy
408	SPINDLE SERIAL LINK START FAULT	This alarm is generated when the spindle control unit is not ready for starting correctly when the power is turned on in the system with the serial spindle. The four reasons can be considered as follows: 1) An improperly connected optic cable, or the spindle control unit's power is OFF. 2) When the NC power was turned on under alarm conditions other than SU-01 or AL-24 which are shown on the LED display of the spindle control unit. In this case, turn the spindle amplifier power off once and perform startup again. 3) Other reasons (improper combination of hardware) This alarm does not occur after the system including the spindle control unit is activated.
409	SPINDLE ALARM DETECTION	A spindle amplifier alarm occurred in a system with a serial spindle. The alarm is indicated as "AL-XX" (where XX is a number) on the display of the spindle amplifier. For details, see Section 14. Setting bit 7 of parameter No. 0397 causes the spindle amplifier alarm number to appear on the screen.

10. ALARM LIST

(7) Over travel alarms

Number	Meaning	Contents and remedy
5n0	OVER TRAVEL : +n	Exceeded the n-th axis + side stored stroke limit 1, 2.
5n1	OVER TRAVEL : -n	Exceeded the n-th axis - side stored stroke limit 1, 2.
5n2	OVER TRAVEL : +n	Exceeded the n-th axis + side stored stroke limit 3.
5n3	OVER TRAVEL : -n	Exceeded the n-th axis - side stored stroke limit 3.
5n4	OVER TRAVEL : +n	Exceeded the n-th axis + side hardware OT. (for M series)
5n5	OVER TRAVEL : -n	Exceeded the n-th axis - side hardware OT. (for M series)
5n4	OVERTRAVEL AT +N AXIS	The tool moved beyond stored stroke limit 4 in the positive direction of the n axis (for T series).
5n5	OVERTRAVEL AT -N AXIS	The tool moved beyond stored stroke limit 4 in the negative direction of the n axis (for T series).
520	OVERTRAVEL AT -Z-AXIS	The tool moved beyond the hardware overtravel position in the positive direction of the Z-axis (for T series).
590	TOOL-POST INTERFERENCE ALARM AT +X-AXIS	A tool-post interference alarm was issued while the tool was moving in the positive direction along the X-axis.
591	TOOL-POST INTERFERENCE ALARM AT -X-AXIS	A tool-post interference alarm was issued while the tool was moving in the negative direction along the X-axis.
592	TOOL-POST INTERFERENCE ALARM AT +Z-AXIS	A tool-post interference alarm was issued while the tool was moving in the positive direction along the Z-axis.
593	TOOL-POST INTERFERENCE ALARM AT -Z-AXIS	A tool-post interference alarm was issued while the tool was moving in the negative direction along the Z-axis.

(8) Macro alarms

Number	Meaning	Contents and remedy
500 to 599	MACRO ALARM	This alarm is related to the custom macro, macro executor, or order-made macro (including conversational program inputs). Refer to the relevant manual for details. (The macro alarm number may coincide with an overtravel alarm number. However, they can be distinguished from each other because the overtravel alarm number is accompanied with the description of the alarm.

(9) PMC alarms

Number	Meaning	Contents and remedy
600	PMC ALARM : INVALID INSTRUCTION	An invalid-instruction interrupt occurred in the PMC.
601	PMC ALARM : RAM PARITY	A PMC RAM parity error occurred.
602	PMC ALARM : SERIAL TRANSFER	A PMC serial transfer error occurred.
603	PMC ALARM : WATCHDOG	A PMC watchdog timer alarm occurred.
604	PMC ALARM : ROM PARITY	A PMC ROM parity error occurred.
605	PMC ALARM : OVER STEP	The maximum allowable number of PMC ladder program steps was exceeded.
606	PMC ALARM : I/O MODULE ASSIGNMENT	The assignment of I/O module signals is incorrect.
607	PMC ALARM : I/O LINK	An I/O link error occurred. The details are listed below.

Number	Details of PMC alarm (No. 607)
010	* Communication error (SLC (master) internal register error)
020	* An SLC RAM bit error occurred (verification error).
030	* An SLC RAM bit error occurred (verification error).
040	No I/O unit has been connected.
050	32 or more I/O units are connected.
060	* Data transmission error (no response from the slave)
070	* Communication error (no response from the slave)
080	* Communication error (no response from the slave)
090	An NMI (for other than alarm codes 110 to 160) occurred.
130	* An SLC (master) RAM parity error occurred (detected by hardware).
140	* An SLC (slave) RAM parity error occurred (detected by hardware).
160	* SLC (slave) communication error * AL0 : Watchdog timer DO clear signal received * IR1 : CRC or framing error Watchdog timer alarm Parity error

Hardware errors are indicated with an asterisk (*).

(10) Overheat alarms

Number	Meaning	Contents and remedy
700	OVERHEAT: CONTROL UNIT	Control unit overheat Check that the fan motor operates normally, and clean the air filter.
704	SPINDLE OVERHEAT	Spindle overheat was detected by the spindle-speed fluctuation detection function (for T series). Check the machining conditions.

10

10. ALARM LIST

(11) M-NET alarm

Number	Meaning	Contents and remedy
899	M-NET INTER-FACE ALARM	This alarm is related to a serial interface for an external PLC. The details are listed below.

Number	Details of M-NET alarm (No. 899)
0001	Abnormal character (character other than transmission codes) received
0002	"EXT" code error
0003	Connection time monitor error (parameter No. 0464)
0004	Polling time monitor error (parameter No. 0465)
0005	Vertical parity or framing error detected
0257	Transmission time-out error (parameter No. 0466)
0258	ROM parity error
0259	Overrun error detected
Others	CPU interrupt detected

(12) System alarms

(These alarms cannot be reset with reset key.)

Number	Meaning	Contents and remedy
910	MAIN RAM PARITY	This RAM parity error is related to low-order bytes. Replace the memory PC board.
911	MAIN RAM PARITY	This RAM parity error is related to high-order bytes. Replace the memory PC board.
912	SHARED RAM PARITY	This parity error is related to low-order bytes of RAM shared with the digital servo circuit. Replace the axis control PC board.
913	SHARED RAM PARITY	This parity error is related to high-order bytes of RAM shared with the digital servo circuit. Replace the axis control PC board.
914	SERVO RAM PARITY	This is a local RAM parity error in the digital servo circuit. Replace the axis control PC board.
915	LADDER EDITING CASSETTE RAM PARITY	This RAM parity error is related to low-order bytes of the ladder editing cassette. Replace the ladder editing cassette.
916	LADDER EDITING CASSETTE RAM PARITY	This RAM parity error is related to high-order bytes of the ladder editing cassette. Replace the ladder editing cassette.
920	WATCHDOG ALARM	This is a watchdog timer alarm or a servo system alarm for axis 1 to 4. Replace the master or axis control PC board.
921	SUB CPU WATCHDOG ALARM	This is a watchdog timer alarm related to the sub-CPU board or a servo system alarm for axis 5 or 6. Replace the sub-CPU board or the axis-5/6 control PC board.
922	7/8 AXIS SERVO SYSTEM ALARM	This is a servo system alarm related to axis 7 or 8. Replace the axis-7/8 control PC board.
930	CPU ERROR	This is a CPU error. Replace the master PC board.
940	PC BOARD INSTALLATION ERROR	PC board installation is incorrect. Check the specification of the PC board.
941	MEMORY PC BOARD CONNECTION ERROR	The memory PC board is not connected securely. Ensure that the PC board is connected securely.

Number	Meaning	Contents and remedy
945	SERIAL SPINDLE COMMUNICATION ERROR	The hardware configuration is incorrect for the serial spindle, or a communication alarm occurred. Check the hardware configuration of the spindle. Also ensure that the hardware for the serial spindle is connected securely.
946	SECOND SERIAL SPINDLE COMMUNICATION ERROR	Communication is impossible with the second serial spindle. Ensure that the second serial spindle is connected securely.
950	FUSE BLOWN ALARM	A fuse has blown. Replace the fuse (+24E; F14).
960	SUB CPU ERROR	This is a sub-CPU error. Replace the sub-CPU PC board.
998	ROM PARITY	This is a ROM parity error. Replace the ROM board in which the error occurred.

(13) External alarm

Number	Meaning	Contents and remedy
1000	EXTERNAL ALARM	This alarm was detected by the PMC ladder program. Refer to the relevant manual from the machine builder for details.

(14) Alarms Displayed on spindle Servo Unit

Alarm No.	Meaning	Description	Remedy
"A" display	Program ROM abnormality (not installed)	Detects that control program is not started (due to program ROM not installed, etc.)	Install normal program ROM
AL-01	Motor overheat	Detects motor speed exceeding specified speed excessively.	Check load status. Cool motor then reset alarm.
AL-02	Excessive speed deviation	Detects motor speed exceeding specified speed excessively.	Check load status. Reset alarm.
AL-03	DC link section fuse blown	Detects that fuse F4 in DC link section is blown (models 30S and 40S).	Check power transistors, and so forth. Replace fuse.
AL-04	Input fuse blown. Input power open phase.	Detects blown fuse (F1 to F3), open phase or momentary failure of power (models 30S and 40S).	Replace fuse. Check open phase and power supply regenerative circuit operation.
AL-05	Control power supply fuse blown	Detects that control power supply fuse AF2 or AF3 is blown (models 30S and 40S).	Check for control power supply short circuit. Replace fuse.
AL-07	Excessive speed	Detects that motor rotation has exceeded 115% of its rated speed.	Reset alarm.
AL-08	High input voltage	Detects that switch is flipped to 200 VAC when input voltage is 230 VAC or higher (models 30S and 40S).	Flip switch to 230 VAC.

10. ALARM LIST

Alarm No.	Meaning	Description	Remedy
AL-09	Excessive load on main circuit section	Detects abnormal temperature rise of power transistor radiator.	Cool radiator then reset alarm.
AL-10	Low input voltage	Detects drop in input power supply voltage.	Remove cause, then reset alarm.
AL-11	Overvoltage in DC link section	Detects abnormally high direct current power supply voltage in power circuit section.	Remove cause, then reset alarm.
AL-12	Overcurrent in DC link section	Detects flow of abnormally large current in direct current section of power circuit.	Remove cause, then reset alarm.
AL-13	CPU internal data memory abnormality	Detects abnormality in CPU internal data memory. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL-15	Spindle switch/output switch alarm	Detects incorrect switch sequence in spindle switch/output switch operation.	Check sequence.
AL-16	RAM abnormality	Detects abnormality in RAM for external data. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL-18	Program ROM sum check error	Detects program ROM data error. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL-19	Excessive U phase current detection circuit offset	Detects excessive U phase current detection circuit offset. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL-20	Excessive V phase current detection circuit offset	Detects excessive V phase current detection circuit offset. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL-24	Serial transfer data error	Detects serial transfer data error (such as NC power supply turned off, etc.)	Remove cause, then reset alarm.
AL-25	Serial data transfer stopped	Detects that serial data transfer has stopped.	Remove cause, then reset alarm.
AL-26	Disconnection of speed detection signal for Cs contouring control	Detects abnormality in position coder signal (such as unconnected cable and parameter setting error).	Remove cause, then reset alarm.
AL-27	Position coder signal disconnection	Detects abnormality in position coder signal (such as unconnected cable and adjustment error).	Remove cause, then reset alarm.
AL-28	Disconnection of position detection signal for Cs contouring control	Detects abnormality in position detection signal for Cs contouring control (such as unconnected cable and adjustment error).	Remove cause, then reset alarm.
AL-29	Short-time overload	Detects that overload has been continuously applied for some period of time (such as restraining motor shaft in positioning).	Remove cause, then reset alarm.

Alarm No.	Meaning	Description	Remedy
AL-30	Input circuit over-current	Detects overcurrent flowing in input circuit.	Remove cause, then re-set alarm.
AL-31	Speed detection signal disconnection motor restraint alarm or motor is clamped.	Detects that motor cannot rotate at specified speed or it is detected that the motor is clamped. (but rotates at very slow speed or has stopped). (This includes checking of speed detection signal cable.)	Remove cause, then re-set alarm.
AL-32	Abnormality in RAM internal to LSI for serial data transfer. This check is made only when power is turned on.	Detects abnormality in RAM internal to LSI for serial data transfer. This check is made only when power is turned on.	Remove cause, then re-set alarm.
AL-33	Insufficient DC link section charging	Detects insufficient charging of direct current power supply voltage in power circuit section when magnetic contactor in amplifier is turned on (such as open phase and defective charging resistor).	Remove cause, then re-set alarm.
AL-34	Parameter data setting beyond allowable range of values	Detects parameter data set beyond allowable range of values.	Set correct data.
AL-35	Excessive gear ratio data setting	Detects gear ratio data set beyond allowable range of values.	Set correct data.
AL-36	Error counter overflow	Detects error counter overflow.	Correct cause, then reset alarm.
AL-37	Speed detector parameter setting error	Detects incorrect setting of parameter for number of speed detection pulses.	Set correct data.
AL-39	Alarm for indicating failure in detecting 1-rotation signal for Cs contouring control	Detects 1-rotation signal detection failure in Cs contouring control.	Make 1-rotation signal adjustment. Check cable shield status.
AL-40	Alarm for indicating 1-rotation signal for Cs contouring control not detected	Detects that 1-rotation signal has not occurred in Cs contouring control.	Make 1-rotation signal adjustment.
AL-41	Alarm for indicating failure in detecting position coder 1-rotation signal.	Detects failure in detecting position coder 1-rotation signal.	Make signal adjustment for signal conversion circuit. Check cable shield status.
AL-42	Alarm for indicating position coder 1-rotation signal not detected	Detects that position coder 1-rotation signal has not issued.	Make 1-rotation signal adjustment for signal conversion circuit.





































10. ALARM LIST

Alarm No.	Meaning	Description	Remedy
AL-43	Alarm for indicating disconnection of position coder signal for differential speed mode	Detects that main spindle position coder signal used for differential speed mode is not connected yet (or is disconnected).	Check that main spindle position coder signal is connected to connector CN12.
AL-46	Alarm for indicating failure in detecting position coder 1-rotation signal in thread cutting operation.	Detects failure in detecting position coder 1-rotation signal in thread cutting operation.	Make 1-rotation signal adjustment for signal conversion circuit Check cable shield status.
AL-47	Position coder signal abnormality	Detects incorrect position coder signal count operation.	Make signal adjustment for signal conversion circuit. Check cable shield status.
AL-48	Position coder 1-rotation signal abnormality	Detects that occurrence of position coder 1-rotation signal has stopped.	Make 1-rotation signal adjustment for signal conversion circuit.
AL-49	The converted differential speed is too high.	Detects that speed of other spindle converted to speed of local spindle has exceeded allowable limit in differential mode.	Check the position coder state of the other side.
AL-50	Excessive speed command calculation value in spindle synchronization control	Detects that speed command calculation value exceeded allowable range in spindle synchronization control.	Check parameters such as a position gain.
AL-51	Undervoltage at DC link section	Detects that DC power supply voltage of power circuit has dropped (due to momentary power failure or loose contact of magnetic contactor).	Remove cause, then reset alarm.
AL-52	ITP signal abnormality I	Detects abnormality in synchronization signal (ITP signal) with CNC (such as loss of ITP signal).	Remove cause, then reset alarm.
AL-53	ITP signal abnormality II	Detects abnormality in synchronization signal (ITP signal) with CNC (such as loss of ITP signal).	Remove cause, then reset alarm.

Alarm No.	Meaning	Description	Remedy
AL-54	Overload current alarm	Detects that excessive current flowed in motor for long time.	Check if overload operation or frequent acceleration/deceleration is performed.
AL-55	Power line abnormality in spindle switching/output switching	Detects that switch request signal does not match power line status check signal.	Check operation of magnetic contractor for power line switching. Check if power line status check signal is processed normally.

11. CORRESPONDENCE BETWEEN ENGLISH KEY AND SYMBOLIC KEY

Table : Correspondence between English key and Symbolic key

Name	English key	Symbolic key
RESET key		
PAGE UP key		
PAGE DOWN key		
SHIFT key		
POSITION key		
PROGRAM key		
MENU key		
OFFSET key		
MENU/OFFSET key		
DIAGNOS/PARAM key		
OPERATION/ALARM key		
AUXILIARY/GRAPHIC key		
MACRO key		
ALTER key		
INSERT key		
INPUT key		
OUTPUT/START key		
DELETE key		
CANCEL key	