Digitized Automation for a Changing World

Delta CNC Machine Solution

NC5 Series– G Command Guidelines

www.deltaww.com

Table of Contents

Preface

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

This manual includes:

- G code instructions and format description
- Macro program and variables definition
- Alarm description

Features of NC5 series controllers

- Built-in 32-bit highspeed dual CPU for multi-task execution and performance improvement
- Friendly HMI Interface
- Auto tuning interface are provided for optimizing the machine's performance efficiency
- CNC Soft software tools to facilitate the development of customized images
- Spindle forms for users to choose between communication type and analog voltage type
- Serial I/O modules for flexible I/O contacts configuration

How to use this manual:

You can consider this manual as a reference for learning to use NC controllers. The manual will tell you how to write G code instructions and how to use variables and MACRO syntax of the NC system. Please be sure to read this manual before starting to use and set up.

DELTA technical services

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

Safety Precautions

- Please follow the instruction of pin definition when wiring. Ground is a must.
- When the power is being supplied, do not disconnect the controller, change the wiring, or touch the power supply.

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

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

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

Installation

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

Wiring

- Connect the ground terminals to a Class 3 ground system. Ground resistance should not exceed 100 Ω. Improper grounding may result in communication error, electric shock, or fire.

Operation

- Use the MLCEditor software to correctly configure the I/O functions, or it may cause abnormal operation.
- Properly set the parameters before operating the machine, or it may cause abnormal operation or malfunction.
- Ensure the emergency stop works properly and avoid operating the machine without protection.

- Do not change the wiring when the power is on, or it may cause electric shock or personnel injury.

Maintenance and Inspection

- Do not touch the internal part of the controller when the power is on, or it may cause electric shock.
- Do not touch the wiring terminals within 10 minutes after turning off the power, or the residual voltage may cause electric shock.
- Cut off the power before replacing the backup battery. Ensure to check the system settings again after replacing the battery.
- Do not block the ventilation holes when operating the controller, or poor heat dissipation may lead to controller malfunction.

Wiring Method

- Power: connect a 24 V_{DC} power to the controller and do the wiring according to the specifications to avoid danger.
- Wire selection: use stranded wires and multi-core shielded-pair wires for all signal cables.
- The local I/O and remote I/O of the controller require an external 24 V_{DC} power supply to output and input signals normally.

Wiring of Communication Circuit

- Ensure the wiring between the controller and the servo drive is firmly connected, or loose connection may result in abnormal operation.

For the differences among the various versions, please refer to DELTA's website for the latest information (https://downloadcenter.deltaww.com/en-US/DownloadCenter).

(This page is intentionally left blank.)

1 Introduction

This chapter introduce the NC 5 controller, description of supporting functions and machine types.

1.1 NC5 Introduction

Delta Electronics has been working in the CNC field for more than a decade and has extensive experience in lathes, milling machines, grinders, and other applications. CNC control is one of the key core technologies for precision manufacturing. In recent years, with the development of multi-axis, multi-channels, and factory automation Intelligent manufacturing has become an indispensable part of the machine tool industry, and the NC5 series controller was born.

The NC5 series is a new generation CNC controller built with high-speed bus and combined with Delta's experience in HMI screen editing and PLC logic control to form a highly flexible development platform. In addition to supporting programs for lathes and milling machines, it also incorporates robot ARM control, creating an efficient motion control system integrating multiple channels and configurations into one machine.

File Management 2

This chapter illustrate the file management of the NC5 series so that users can have a better understanding of the file system.

2.1 Folder management

2

This section describes four primary folder types, including INTER, SD card, USB drive and NETDRIVE for internet shared folder.

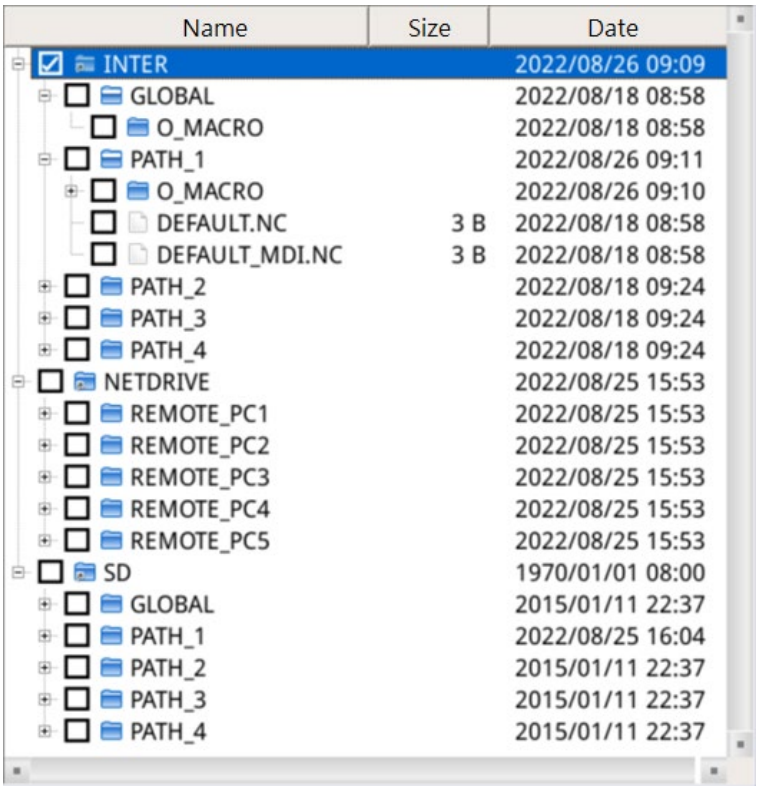


Figure 2.1.1 System default folder

2.1.1 Folder name

Folder names can be named in multiple languages and supports ASCII encoded text input. The maximum name length is limited to 255 characters (Chinese characters occupy two characters).

Attention:

1. Forbidden to use [.] and [..] character, which are reserved for system folder naming.
2. Forbidden to use [:] and [/] character, which are reserved for system directory naming.

2.1.2 Public folder

The GLOBAL folder is shared by the entire public domain and can be accessed by all NC paths. Parameter **[N8.22 Bit11]** can be set to hide all GLOBAL folders. It contains preset O_MACRO for placing system-shared macro files. Please do not remove this directory.

2.1.3 NC path folder

The dedicated PATH_□ folder of the NC path is exclusive to each NC path and cannot be used across NC paths. Parameter **[N8.22 Bit12]** can be set to hide all PATH_□ folders.

It contains system default files:

1. The subdirectory O_MACRO is used to place the exclusive macro files in the NC path. Please do not remove this directory. Parameter **[N8.22 Bit10]** can be set to hide the O_MACRO folder.
2. The default NC path main program is DEFAULT.NC. Please do not remove it.
3. The Default MDI program DEFAULT_MDI.NC of the NC path, please do not remove it.

2.1.4 Relevant parameters

This table lists the relevant parameters used by the archive.

Parameter	Name	Description
N8.20 Bit 5	Enable O Marco file encrypt protection	0: Disable 1: Enable
N8.22 Bit 0	Setting for text editor can be edit by users	0: Allowed. 1: Prohibited.
N8.22 Bit 1~2	Setting for the file resource of macro and sub-program	0: INTER 1: SD card
N8.22 Bit 4	Auto focus after program edit	0: Disable 1: Enable
N8.22 Bit 5	Macro call program file direction	0: Same direction as main program. 1: USB drive.
N8.22 Bit 6	Whether to display the sub-program name when the main program calling	0: Display 1: Hide.
N8.22 Bit 8	Path of friction compensation measurement program	0: O_MACRO 1: SD
N8.22 Bit 10	Whether to display the macro content when the O macro executing	0: Hide 1: Display.
N8.22 Bit 11	Whether to display the Global G code folder	0: Display 1: Hide.
N8.22 Bit 12	Whether to display the Path G code folder	0: Display 1: Hide.

2.2 File management

2.2.1 Main program

2

The main program file can be named in multiple languages and supports ASCII encoded text input. The maximum name length is limited to 255 characters (Chinese characters occupy two characters). Any program file can be set as the main program and the system will keep the main file. If the main program is removed, the system will switch to the default program which is stored in the corresponding NC path folder as the main program (INTER:/PATH_□/DEFAULT.NC).

2.2.2 Subprogram / macro program

Subprogram/ macro program file names are limited to O0001 to O9999, which are used for M98, G65... and other command calls. The file naming rules are as follows:

1. Support without sub-file name or sub-file name with [.NC].
2. O50000 and O60000 need to reserve for main program and MDI program to use.
3. O9000 can be used for T code calling function, which depends on parameter **[N1.10 Bit22]** setting.
4. O9030 can be used for break searching calling function, which depends on parameter **[N1.20 Bit22]** setting.
5. O9100 to O9149 can be used for G code calling function, which depends on parameter **[N1.200] ~ [N1.249]** setting.
6. O9150 to O9199 can be used for M code calling function, which depends on parameter **[N1.250] ~ [N1.299]** setting.
7. O8000 to O9999 can be used for G or M code calling function, which depends on parameter **[N1.120] ~ [N1.125]** setting.
8. N8.22 can set for macro call program file direction, the searching priority as below table:
   Note: The program file direction is defined in the parameter **[N8.22 Bit 1~2]** as (DISK) in the below table, which describe the file search priority.

Attention:

A. Forbidden to use [.] and [..] character, which are reserved for system folder naming.
B. Forbidden to use [:] and [/] character, which are reserved for system directory naming.
C. Avoid to use [\ ], [?], ["], [*], [<], [>], [.] and control type character, because these may let the system block it as illegal file.

parameter	Macro call function and program file search priority	
N8.22 Bit 5 Macro call program file direction	M98 call sub-program. M96 call macro. G65 call macro. G66 call macro.	G, M, T code macro. The macro after break & search. Cycle starts initial macro. One button call macro.
0 Same root as main program	(1) Same root as main program (2) (DISK)/PATH_□ (3) (DISK)/PATH_□/O_MACRO (4) (DISK)/GLOBAL (5) (DISK)/GLOBAL/O_MACRO	(1) (DISK)/PATH_□ (2) (DISK)/PATH_□/O_MACRO (3) (DISK)/GLOBAL (4) (DISK)/GLOBAL/O_MACRO (5) Same root as main program
1 USB root path	(1) USB root path (2) (DISK)/PATH_□ (3) (DISK)/PATH_□/O_MACRO (4) (DISK)/GLOBAL (5) (DISK)/GLOBAL/O_MACRO	(1) USB root path (2) (DISK)/PATH_□ (3) (DISK)/PATH_□/O_MACRO (4) (DISK)/GLOBAL (5) (DISK)/GLOBAL/O_MACRO

2.3 External storages

2.3.1 USB storage

The system will display the connected USB devices and including its folders and files. However, the system will not automatically generate system default folder and files on this type of devices.

Attention:

1. The USB devices must be FAT32 and exFAT format.
2. The USB devices must be set as single sector. If it has multi-sector or hidden sector, the system will not be able to access it.
3. When the main program is store in the external devices and once the devices removed, the system will automatically set the specific DEFAULT.NC as the main program. If users set this DEFAULT.NC as main program manually, the system will reload the last program as main program after system restart.
4. If select the program in the USB device as main program, please do not remove the device when program executing. Otherwise, it may cause the product damaged.

2.3.2 Micro SD(TF) storage

The system will display the connected micro-SD(TF) devices and including its folders and files. The system will automatically generate [Global] system folder and folder [PATH_□] for each NC paths.

Attention:

1. The micro-SD(TF) devices must be FAT32, exFAT and EXT4 format.
2. The micro-SD(TF) devices must be set as single sector. If it has multi-sector or hidden sector, the system will not be able to access it.

2.3.3 NETDRIVE internet share folder

The NC5 controller supports SAMBA communication protocol which allowed users to access the remote shared folder from REMOTE_PC1 to REMOTE_PC5.

Attention:

1. The remote IP address please refers to parameter **[N8.105] ~ [N8.109]**.
2. Please make sure the Ethernet communication quality when executing the main program through the shared file. Otherwise, it may cause the product damaged.

NC Programing 3

This chapter introduces the G-code formats supported by the NC5 series controllers along with application examples, which allows users to learn more about G-code and macro commands.

3

3.1 Axis definition

3

3.1.1 Axis naming rule

The NC5 NC system supports standard axes naming as well as user-defined axis naming, which can be up to three characters. User-defined axis names have two definition configurations, including the interpretive name for NC commands and the display name, as described below.

User-defined interpretive name: axis name used for NC program execution.

(1) After a new interpretive name is configured, the corresponding axis name in the NC program should be also modified.

(2) If the last character of the axis interpretive name is set to 0–9, the program should use the (=) symbol to distinguish the interpretive name and its movement command. For example: XA1=100.

Display Name: the displayed name on the system coordinates.

(1) This is only for displaying the axis name on the system interface and does not affect editing the program.

(2) Set the parameter and then restart the controller, the system will automatically change its axis name display in both the coordinate system and the axis parameter page.

- The 1st character can only be X, Y, Z, A, B, C, U, V, W or is not set.
- The 2nd character can only be A-Z and number 0-9 or is not set.
- The 3rd character can only be A-Z and number 0-9 or is not set.

  If the 2nd character is 0–9, the 3rd character will only accept 0–9 or is not set.

	1st character	2nd character	3rd character
Available setting	X, Y, Z, U, V, W, A, B, C	A~Z	A~Z
	-	0~9	0~9
Correct	Y	1	1
Correct	Y	A	1
Correct	Y	A	B
Wrong	Y	1	A

Note:

(1) Each axis name should be kept unique, otherwise the system will return an alarm for axis name duplication.

(2) When the mechanical type is set to lathe and the Type A parameter **[N1.8 lathe G-code type]** = 0, the NC axis cannot be set to U, V or W; the U, V and W axis name code will keep as incremental input.

(3) The interpretive name and the display name can be setting on the path parameters' page.

Channel		Axis	Enable	Type	SP ID	Port	Serial	Display	InterPret	DisplayName
CH 1		X	■	1		1	1	■	XA1	XAB
		Y	■	1		2	2	■	Y	Y
		Z	■	1		3	3	■	ZA1	ZA1
		A	□							
		B	□							
		C	□							
		U	□							
		V	□							
		W	□							
		AX1	□							
		AX2	□							
		AX3	□							
		SP1	■	3	1	4				
		SP2	□							
Model	MILL	SP3	□							
Enable	■	SP4	□							

3.1.2 Mechanical unit

The NC axis has two types of definitions, rotary axis and linear axis. The unit of the linear axis can be defined in the parameter **[N1.11 Bit2 screw unit setting]** as mm or inch. The rotation axis will always use deg as the control unit.

Note:

(1) The axis type setting is configured in the parameter **[N2.001 Bit2–4 rotation axis feed mode]**.

(2) The NC position command unit will apply 0.001 as the reference distance (0.001 mm or 0.001 inch) if the distance command in the NC program doesn't include a decimal point. Users can configure the **[N8.018 Bit10 the function of automatically dividing thousands without decimal point disabled]** to disable this default function.

(3) The parameter **[N1.020 Bit0 ON initial position]** can be configured in the default NC program unit as mm or inch.

(4) The parameter **[N9.014 system length unit]** can be configured in the system default display unit as mm or inch.

3.2 Block and group instructions

This section lists the G code commands for milling and lathe machines in the NC5 system. In addition, the lathe systems can be divided into A, B, and C types based on the G code command type setting, which is set by the parameter **[N1.08 G code command type in lathe mode]**. The functions and corresponding commands are listed in **[3.2.3 Command table]** and users can find all related descriptions in this manual.

3.2.1 Command block

The NC instructions use a line of text as a command block, and mainly use G, M, S, and T as their constituent elements.

G: Command function

M: Auxiliary function

S: Spindle axis rotation speed

T: Tool selection

G codes are divided into single block instructions and status instructions. The functions of single block instructions are only effective within a single block, and its functions will not be retained in other blocks. Status instructions contain 32 G code commands in the system. A single status instruction only maintains a single status at a time until it is replaced by another set of status instructions.

When the above instructions are used in one single block, the functional principles are as follows:

(1) When applying G, M, or T codes as call macro functions, the command code should be at the beginning of the single block.

(2) When applying G, M, or T codes as call macro functions, other codes in the single block will be regarded as arguments; the value in the argument corresponds to the # variable, please refer to the section **[3.11.1 #macro variable]** for instructions.

(3) For non-calling G code instructions, M, S, and T codes are allowed in the same block as auxiliary functions.

(4) For non-calling M code instructions, S and T codes are allowed in the same block as auxiliary functions.

(5) For non-calling T code instructions, M and S codes are allowed in the same block as auxiliary functions.

3.2.2 Command look-ahead

To perform path interpolation such as acceleration and deceleration of each axis, single block fitting, and path smoothing when the system executes the NC program, the system will apply look-ahead to preview multi-line programs into the memory in order to plan the overall machining path in advance.

However, this function requires special attention to the timing when using functions such as # variables or switching axis control, as explained below. If users encounter related problems, they can pause the preview process through the special M relay to halt the path interpretation:

(1) The program uses the corresponding # variable for MLC interactions.

#Number	Description	Attribute
25000 ~ 25127	Special M update to # variable. ex: M2x128 update to #25000	R
25128 ~ 25255	Special D update to # variable. ex: D2x128 update to #25128	R
25256 ~ 25383	# variable write to special M. ex: #25256 write to M3x128	R/W
25384 ~ 25511	# variable write to special D. ex: #25384 write to D3x128	R/W

(2) The program uses axis switching control functions, for example: axes synchronize, command transfer, NC axis and MLC axis exchange, lathe spindle and C-axis exchange. The preview function will let the system interpolate a path when the axes have not yet been switched.

(3) When the program uses a # variable for calculation, and the calculation result will use process commands such as IF and WHILE for condition judgment, sometimes it will cause unexpected judgments because of the look-ahead preview, thereby causing the # variable calculation to produce unexpected results.

Note:

1. General M codes in the NC program will not stop the NC system to preview the instructions. Please set **[N1.118 Beginning M code of halt function] and [N1.119 Amount of halt M code]** to define the special M for halt function.
2. Some of the G code commands also have the attribution to pause the NC previewing. Please refer to the **[3.2.4 Command table]** section for detailed information.

3.2.3 Command group

3

The following table lists the 32 groups of G code statuses in the NC5 controller. Except for group 00, the G code status instructions belonging to the same group will only have one status at any given time. If the default status settings of parameter **[N1.20 G code application setting]** are all 0, the system will use the codes in bold in the following table as the initial status of the system.

Group	Milling	Lathe		
		A	B	C
00	Single block available			
01	**G00**, G01, G02, G03			
02	**G17**, G18, G19			
03	**G90**, G91		**G90**, G91	**G90**, G91
04	reserved			
05	**G94**, G95	**G98**, G99	**G94**, G95	**G94**, G95
06	G20, **G21**		G70, **G71**	
07	**G40**, G41, G42			
08	G43, G44, **G49**			
09	**G80**, G7X, G8X cycle command			
10	**G98**, G99		G98, G99	G98, G99
11	**G50**, G51			
12	**G54**, G55, G56, G57, G58, G59			
13	G61, **G64**			
14	G66, **G67**			
15	**G68**, G69			
16	**G15**, G16			
17	G96, **G97**			
18	G24, **G25**			
19 | 30	Reserved			
31	G54 extension coordinate system P1 ~ P64			

3.2.4 Command table

Function	Stop preview	Milling		Lathe				page
		groups	Code	A	B	C	groups	
Rapid positioning		01	G00				01	3-12
Linear interpolate			G01					3-13
Clockwise circular interpolation			G02					3-15
Counterclockwise circular interpolation			G03					
3D circular interpolation (3 points command)			G02.4	-				3-18
			G03.4					
Dwell		00	G04				00	3-47
Parameter group change	●		G05					
Exact stop		00	G09				00	3-21
Programmable data input	●		G10					3-43
Programmable data input cancel	●		G11					3-43
Polar coordinates command cancel		16	G15	-			16	3-19
Polar coordinates command			G16					
X-Y plane selection		02	G17				02	3-32
Z-X plane selection			G18					3-32
Y-Z plane selection			G19					3-32
inch input		06	G20	G20	G20	G70	06	3-40
mm input			G21	G21	G21	G71		3-40
Return to 1st reference position		00	G28	G28	G28	G28	00	3-25
Return from reference position			G29	G29	G29	G29		3-26
Return to 2nd, 3rd, 4th reference point			G30	G30	G30	G30		3-27
Skip function	●		G31	G31	G31	G31		3-29
Threading		-		G32	G33	G33	01	3-
Tool radius compensation cancel		07	G40	G40	G40	G40	07	3-76
Tool compensation left			G41	G41	G41	G41		
Tool compensation right			G42	G42	G42	G42		
Tool length positive compensation		08	G43	-				3-81
Tool length negative compensation			G44					
Tool length compensation cancel			G49					
Scaling cancel		11	G50	-	G50	G50	11	3-83
Scaling			G51		G51	G51		

3

Functions	Stop preview	Milling		Lathe				page
		Group	code	A	B	C	Group	
Coordinate system setting		00	G92	G50	G92	G92	00	3-37
Max spindle speed clamp								
Mirror cancel		18	G25	G50.1	G50.1	G50.1	18	3-84
Mirror enable			G24	G51.1	G51.1	G51.1		
Local coordinate system			G52					3-33
Machine coordinate system	●		G53					
1st workpiece coordinate selection		12	G54				12	3-35
2nd workpiece coordinate selection			G55					
3rd workpiece coordinate selection			G56					
4th workpiece coordinate selection			G57					
5th workpiece coordinate selection			G58					
6th workpiece coordinate selection			G59					
Exact stop mode		13	G61				13	3-22
Cutting mode			G64					
Macro call		00	G65				00	3-67
Macro modal call		14	G66				14	3-69
Macro modal call cancel			G67					
Coordinate rotation		15	G68	-				3-85
Coordinate rotation cancel			G69					
Peck drilling cycle		09	G73	-				3-86
Left hand tapping cycle			G74					3-91
Fine boring cycle			G76					3-93
Canned cycle cancel			G80					3-98
Drilling cycle			G81					3-86
Drilling with dwell			G82					3-86
Deep hole peck drilling cycle			G83					3-86
Right hand tapping cycle			G84					3-91
Reaming cycle			G85					3-93
Rough boring cycle			G86					3-93
Back boring cycle			G87					3-93
Boring cycle with dwell			G88					3-93
Back boring cycle with dwell			G89					3-93

Functions	Stop preview	Milling		Lathe				page
		Group	code	A	B	C	Group	
Finishing cycle		-		G70	G70	G72	09	3-105
Stock removal cycle in turning				G71	G71	G73		3-106
Stock removal cycle in facing				G72	G72	G74		3-108
Profile rough turning cycle				G73	G73	G75		3-111
End face peck drilling cycle				G74	G74	G76		3-113
Outer/internal diameter drilling cycle				G75	G75	G77		3-115
Multiple-thread cutting cycle				G76	G76	G78		3-126
Outside/inside diameter cutting cycle				G90	G77	G20		3-118
Threading cycle				G92	G78	G21		3-123
End face turning cycle				G94	G79	G24		3-116
Canned cycle cancel				G80	G80	G80		3-104
End face drilling cycle				G83	G83	G83		3-100
End face tapping cycle				G84	G84	G84		3-101
End face boring cycle				G85	G85	G85		3-103
Side drilling cycle				G87	G87	G87		3-100
Side tapping cycle				G88	G88	G88		3-101
Side boring cycle				G89	G89	G89		3-103
Absolute command		03	G90	-	G90	G90	03	3-38
Increment command			G91		G91	G91		3-38
Feed per minute (mm/min)		05	G94	G98	G94	G94	05	3-24
Feed per rotation (mm/rev)			G95	G99	G95	G95		
Constant linear speed control (m/min)		17	G96	G96	G96	G96	17	3-50
Constant linear speed control cancel (rev/min)			G97	G97	G97	G97		
Return to initial point in canned cycle		10	G98		G98	G98	10	11
Return to R point in canned cycle			G99		G99	G99		
MLC M relay control		00	G900	-	-	-	00	3-48

3.3 Interpolation function

3

3.3.1 Rapid traverse (G00)

Rapid traverse		group	Command code
Milling		01	G00
Signal machine	A-Type		
	B-Type		
	C-Type		

Instruction format:

G00	CP

CP the target position in the absolute mode.
the movement distance in the increment mode.

Instruction description:

Rapidly move the tool to the command position.

Feeding speed:

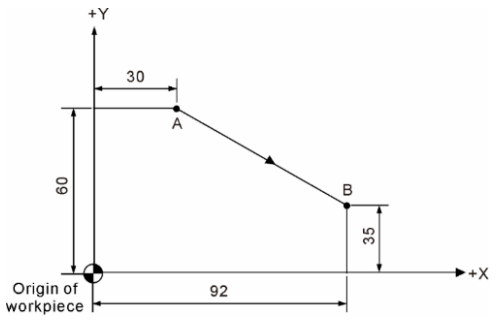
The operation mode of G00 can be divided into two types of speed through the setting of parameter **[N1.20 bit18]**. When the parameter is set to 0, the tool will move linearly at rapid speed, thus all the command axes will move at a composite speed, which is determined by **[N1.30]** – **[N1.32]**. When the parameter is set to 1, the system will move the axes independently with the speeds defined by parameters **[N2.020]** – **[N2.022]**.

Notes:

(1) Cutting speed is controlled by D2x004 as a rapid feed override.

(2) In A-Type lathe machines, the U, V and W represent relative coordinates.

Example:

Example	Program/ instruction
	Rapidly positioning to B point from current position A: Absolute command expression: G90 G00 X92. Y35. Incremental command expression: G91 G00 X62. Y-25.

3.3.2 Linear interpolate (G01)

Linear interpolate		Group	Command code
Milling		01	G01
Lathe	A-Type		
	B-Type		
	C-Type		

Command format:

G01	CP	F_

G01	CP	F_	,	C_
				R_
				A_

CP The target position in the absolute mode.
The movement distance in the increment mode.

F_ Feed speed, it is continuously valid until it is replaced by a new command value.

C_ Sets the chamfer length value between two lines when they are at a 45-degree angle.

R_ Set fillet radius for two lines connecting in an arc.

A_ Set the specific connect degree value between two lines.

Instruction description:

Move the tool linearly from the current position to the specified coordinate position at cutting speed. The cutting speed specified by the F command. if not specified, the system will refer to parameter **[N1.042 Default cutting feed]**.

Automatic chamfering:

C_, R_ and A_ are used as the selection argument and should be entered after the comma symbol. The function is to produce the chamfer between two linear commands, which helps simplify the chamfer path programming. Please refer to parameter **[N1.013 Bit 5 Chamfer and fillet format setting]**.

Note:

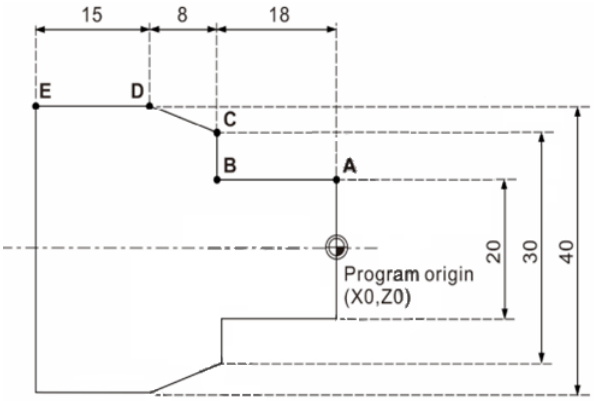
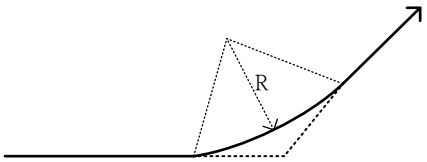
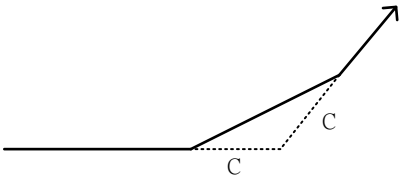
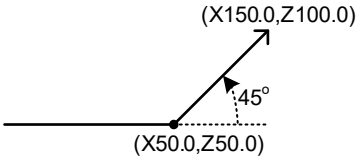
(1) If the cutting speed is not specified, the system will refer to parameter **[N1.042 Default cutting feed]**.

(2) Cutting speed is affected by **D2X002** as feed override.

(3) In the A-Type lathe machines, U_, V_, W_ represent the relative coordinates of the X, Y, Z axes.

3

Example description:

Example	Program/ Instruction
	G98 G54 X0.0 Z0.0 G00 X20.0 G01 Z-18. F500 X30. X40. Z-26. Z-41. M30
	(Two linear paths connected with fillet arc) G91 G01 X100.0,R10. X100.0 Y100.0
	(Two linear paths connected with chamfer) G91 G01 X100.0 , C10. X100.0 Y100.0
	(Two linear paths connected with specific degree angle) G90 G00 X50.0 Z50.0 G01 Z100.0,A45.0

3.3.3 Circular interpolate (G02/G03)

Clockwise circular interpolation		Group	Command code
Milling		01	G02
Lathe	A-Type		
	B-Type		
	C-Type		

Counterclockwise circular interpolation		Group	Command code
Milling		01	G03
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G17	G02 G03	CP	I_J R_	F_
G18	G02 G03	CP	I_K_ R_	F_
G19	G02 G03	CP _	J_K_ R_	F_

CP The target position in the absolute mode.
The movement distance in the increment mode.

R _ Arc radius.
(It is called the radius rule when represented by R)

I_ The relative X axis distance from the start position to the center of the circle.

J_ The relative Y axis distance from the start position to the center of the circle.

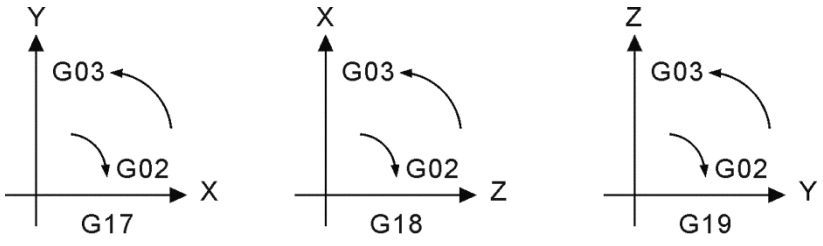
K_ The relative Z axis distance from the start position to the center of the circle.
(It is called center mode when represented by I, J and K)

F_ Feed speed, it is continuously valid until it is replaced by a new value.

Instruction description:

◆ **Arc direction definition:**

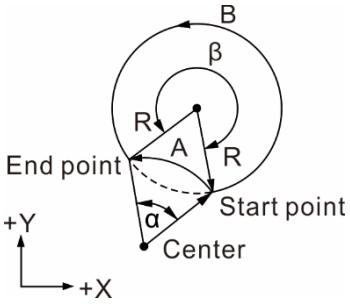
As shown below, on the two-dimensional plane of the standard Cartesian coordinate system, clockwise movement is G02 and counterclockwise movement is G03.



3

◆ **Radius rule**

The radius rule uses the R argument to specify the arc radius. This method uses the starting point, end point, and arc radius to represent the arc path, but two possible sets of arc paths can be generated based on these conditions, as shown in the paths A and B in the figure below. Therefore, the system will select the path based on the positive and negative values of the argument R. When R is a positive value, the arc path with the central angle ≤180° will be selected; if it is a negative value, the arc path with the central angle > 180° will be selected.



If the center degree of arc A is a < 180, the R command value with use the negative value.

If the center degree of arc B is β > 180, the R command value will use the positive value.

In the above figure, if R=50mm and the end-point absolute position is (100.0, 80.0), then:

The center degree of the arc > 180°.

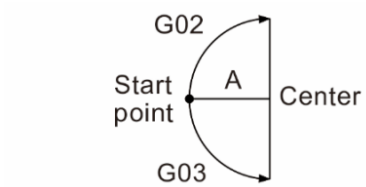
(Path B) G90 G03 X100.0 Y80.0 R-50.0 F80

The center degree of the arc ≤ 180°.

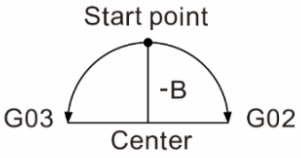
(Path A) G90 G03 X100.0 Y80.0 R50.0 F80

◆ **Center rule**

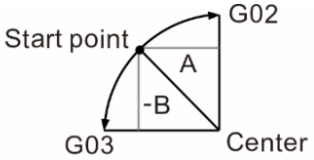
The center rule uses I, J, and K arguments to describe the relative distance from the starting point to the center of the arc in the X, Y, and Z directions, as illustrated in the following figure. When users want to program a closed circular path, it is more appropriate to use the center rule.



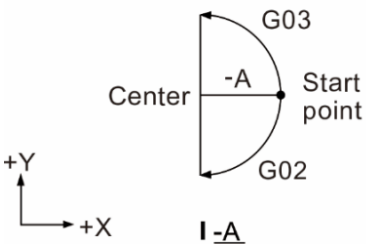
I A



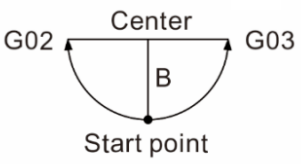
J -B



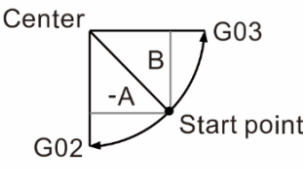
I A J -B



I -A



J B



I -A J B

Note:

(1) The G17 (X-Y plane) is the system default plane. Therefore, when working only on the X-Y plane, the G17 instruction can be omitted.

(2) If the I, J and R arguments are programed in the same block, the system will use the R argument only as the radius rule. The I and J arguments will be ignored.

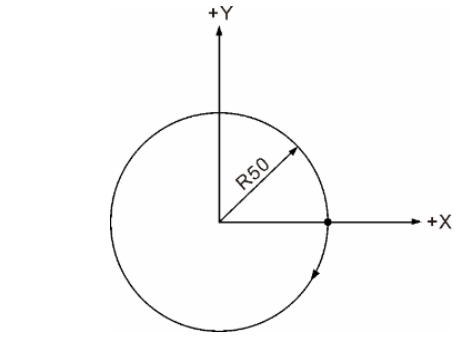
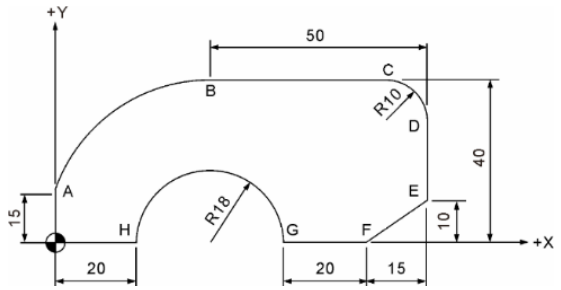
(3) I, J, and R arguments can be omitted when the value is 0, such as I0, J0, or K0.

(4) When the X, Y, and Z end point coordinates are omitted in the block, it will be considered as the starting point and the end point being at the same position, which is a closed circular path; if the radius rule is used, there will be no tool path generated.

(5) When the distance between the end point coordinates and the calculated circle center is not equal to the command radius or exceeds the parameter set in **[N1.040 Arc command radius tolerance]**, the warning message "Arc Radius Error" will be displayed.

(6) When the R, I, J, K arguments in the arc instruction (G02, G03) block are not specified, the path result will be the same as the G01 instruction.

Example:

Example	Program/ instruction
	Closed circle path milling: G02 I-50.0
	Combination command: G90 G54 X0 Y0 S500 M3 (Original point) G90 G01 Y15.0 F80 (A point) G02 X41.0 Y40.0 R41.0 (B point) G91 G01 X40.0 (C point) G02 X10.0 Y-10.0 R10.0 (D point) G01 Y-20.0 (E point) X-15.0 Y-10.0 (F point) X-20. (G point) G90 G03 X20.0 R18.0 (H point) G01 X0.0 (Original point)

3

3.3.4 3D circular interpolation (G2.4/G3.4)

3D circular interpolation (3 points forming a circle)		Group	Command code
Milling		01	G2.4/G3.4
Lathe	A-Type	-	
	B-Type		
	C-Type		

Instruction format:

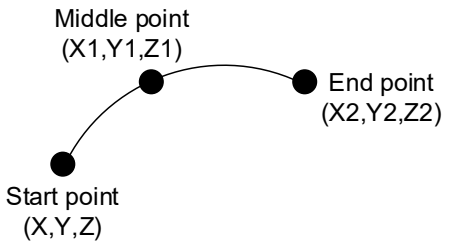
	Command	LP	Description
Format 1	G2.4	X_1 Y_1 Z_1	(Middle-point position)
	G2.4	X_2 Y_2 Z_2	(End-point position)
Format 2	G3.4	X_1 Y_1 Z_1	(Middle-point position)
	G3.4	X_2 Y_2 Z_2	(End-point position)

X_1 Y_1 Z_1 The middle position of the circle command in absolution mode.
The relative distance from the start position to the middle position of the circle in relative mode.

X_2 Y_2 Z_2 The end position of the circle command in absolution mode.
The relative distance from the start position to the end position of the circle in relative mode.

Instruction description:

This command uses the theory of three points forming a circle and program two follow-up single block commands of G2.4/G3.4 to define two coordinate points in the space. The first command is the middle point, and the second command is the end point. As shown in the figure below, the current coordinates are used as the starting point (X, Y, Z), and the arc in the 3D space can be determined by specifying the middle point (X_1, Y_1, Z_1) and the end point (X_2, Y_2, Z_2).



Notes:

(1) When the is G91 relative command is applying, the middle point (X_1, Y_1, Z_1) is relative to the current position, and the end point (X_2, Y_2, Z_2) is relative to the middle point (X_1, Y_1, Z_1).

(2) The closed circle path is not supported.

(3) Please disable the G41 or G42 compensation before using 3D circular interpolation instruction.

3.3.5 Polar coordinates command (G15/G16)

Polar coordinates command cancel		Group	Command code
Milling		01	G15
Lathe	A-Type	-	
	B-Type		
	C-Type		

Polar coordinates command		Group	Command code
Milling		01	G16
Lathe	A-Type	-	
	B-Type		
	C-Type		

Instruction format:

G15
G16

Instruction description:

The polar coordinates are a set of commands that only support the G code 16 group in the milling machine system. All motion commands are based on a rotation center, while the path is planned to use the defined radius and angle.

When this mode is enabled, the plane selection of the G code 02 group will be used to determine the code used in subsequent instructions to define the radius and angle of the target coordinates. The details are as follows:

- G17: X-Y plane, X_ specifies the radius, Y_ specifies the angle.
- G18: X-Z plane, Z_ specifies the radius, X_ specifies the angle.
- G19: Y-Z plane, Y_ specifies the radius, Z_ specifies the angle.

Both the radius and angle commands of polar coordinates support absolute value and relative value commands:

Absolute command: The radius command is the distance from the rotation center to the origin of the working coordinate system. If the local coordinate system G52 is enabled, the origin of the local coordinate system will be used as the reference point.

The angle command is the angle relative to the reference axis of the specified plane.

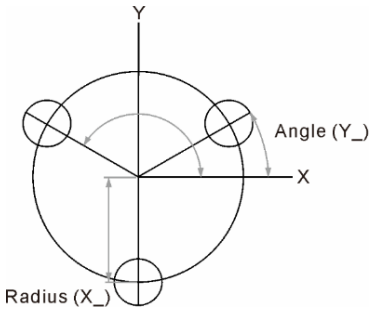
Relative command: The radius command is the distance from the rotation center to the current position.

The angle command is an increment degree that is added on to the previous command.

Execute the G15 command to cancel the polar coordinate function. After canceling the polar coordinate function, the program will resume its motion path with the original coordinate system and coordinate values.

3

Example:

Example	Program/ Instruction
	Polar coordinate for drilling: G90 G16 (Enable polar coordinate) G81X100.0Y30.0Z-20.0R-5.0F200.0 (R100,030° drilling) X100.0Y150.0 (R100,150° drilling) X100.0Y270.0 (R100,270° drilling) G15G80 (Cancel polar coordinate) M30

3.4 Feed function

3.4.1 Exact stop (G09)

Exact stop		Group	Command code
Milling		00	G09
Lathe	A-Type		
	B-Type		
	C-Type		

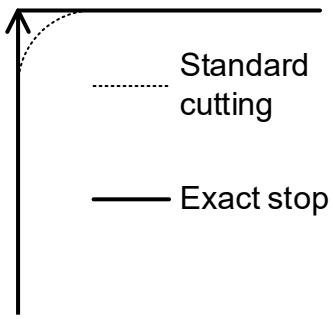
Instruction format:

G09	G01	CP
	G02	
	G03	

CP The target position in the absolute mode.
The movement distance in the increment mode.

Instruction description:

When executing path processing, in order to keep the feed speed continuous, the system will execute the next block in advance before the execution of the block is completed so that the running speed can be continued. This continuous motion will produce rounded corners on the path. It will also prevent the running path from reaching the target position. If users need to move to the target position, they can use the G09 command to actually finish the motion block and the system will detect the tool positioning. The system will execute the next block command until the positioning is confirmed. The description is shown below:



Note:

(1) This command is only valid for cutting commands (G01 – G03) in a single block.

(2) When using this exact stop instruction, it will lead to the speed not being continuous between each motion block.

3

3.4.2 Exact stop mode (G61)

Exact stop mode		Group	Command code
Milling		13	G61
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G61

Instruction description:

The G61 instruction functions similarly to the G09 instruction. The difference is that the G09 instruction is not a continuously valid instruction, it is only valid at the single clock. The G61 instruction is a continuously valid instruction. After specifying the G61 instruction, every G01, G02, G03 command will execute exact stop until the system meets the G64 cutting mode instruction. Otherwise, this mode status will remain valid.

3.4.3 Cutting mode (G64)

Cutting mode		Group	Command code
Milling		13	G64
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G64

Instruction description:

G64 is the system default mode. In this mode, the system will apply a continuous speed between each motion block instruction, so that the end point of each movement instruction will not decelerate to zero and the target position will also not be reached.

Note:

After executing the G64 command, the system will still decelerate to zero speed and check the target position for path planning when encountering the following situations:

- G00 rapid traverse instruction.
- G09 exact stop instruction.
- The next command block has no movement.

3

3.4.4 Cutting feed mode (G94/G95/G98/G99)

Feed per minute (mm/min)		Group	Command code
Milling		05	G94
Lathe	A-Type		G98
	B-Type		G94
	C-Type		G94

Feed per rotation (mm/rev)		Group	Command code
Milling		05	G95
Lathe	A-Type		G99
	B-Type		G95
	C-Type		G95

Milling instruction format:

G94	F_
G95	

G94F_ Cutting feed speed per minute, unit: mm/min or inch/min.

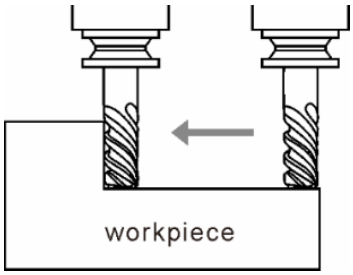
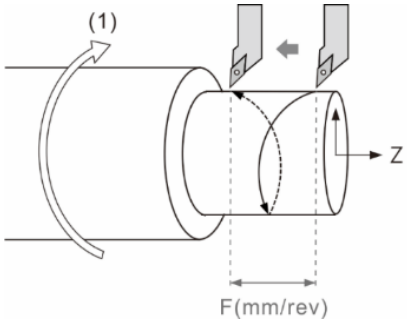
G95F_ Cutting feed speed per spindle revolution, unit: mm/rev or inch/rev.

Instruction description:

The G code group 05 includes two types of speed control modes, as described below:

- The tool feed distance each minute, unit: mm/min or inch/min.
- The tool feed distance each spindle revolution, unit: mm/rev or inch/rev.

This command is a continuously valid command and can be programed in the same block with the motion command or separately in a single block.

Example	Program/ Instruction
	Tool process distance per minute: This mode is not related to spindle rotate speed.
	Tool process distance per spindle revolution: (Lathe spindle) M3S100 (Spindle positive rotate 100RPM) G99 G01 Z-20. F0.35 (Feed 0.35 mm/rev) Actual cutting speed is: 100 rev/min * 0.35 mm/rev = 35.000 mm/min

3.5 Reference position

3.5.1 Return to 1st reference position (G28)

Return to 1st reference position		Group	Command code
Milling		00	G28
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G28	CP

CP coordinate value of intermediate point.

Instruction description:

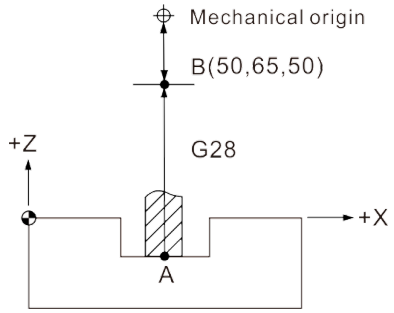
The G28 command will use the G00 rapid positioning method to return to the 1st reference point through the intermediate point set by the command. The purpose is to allow the tool to quickly return to the 1st reference point and avoid machine interference problems through the intermediate point. The CP in the command format is expressed as the intermediate position. The axis whose intermediate coordinate is not specified will directly return to the 1st reference point.

When tool G41 or G42 radius compensation is set, it is recommended to cancel the tool radius compensation function first. If the tool radius compensation function is not canceled, the system will move to the middle point and return to the 1st reference point using a path without tool radius compensation when executing the G28 action. Then, when the next motion block is activated, the system will automatically restore the tool radius compensation function. In addition, the tool length compensation function of G43 or G44 will still be valid before reaching the intermediate point, and then it will return to the 1st reference point without tool compensation when G28 is executed. The tool length compensation will be canceled and will not automatically recover anymore, users need to give the tool length compensation command again.

Note:

The 1st reference point is set in parameter **[N2.060 1st reference coordinate]**.

Example description:

Example	Program/ Instruction
	1st reference point return G90 G28 Z50. (The Z axis returns to the first reference point from point A through point B) M06 T02 (Exchange to tool NO.2)

3

3.5.2 Positioning from reference position (G29)

Return from reference position		Group	Command code
Milling		00	G29
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G29	CP

CP Target position

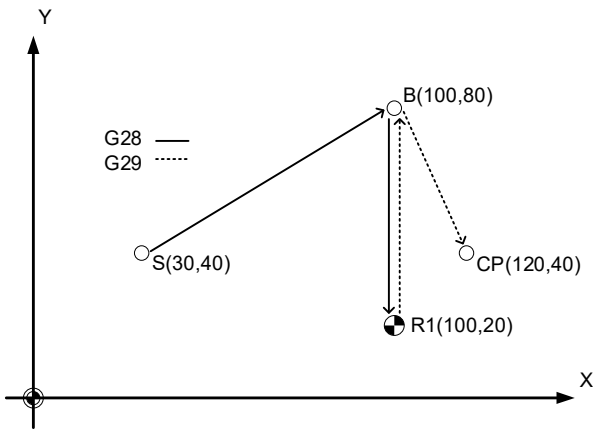
Instruction description:

The G29 command rapidly positions the tool from the current position to the reference point G28 or G30, and then moves to the CP target position specified in the block. The reference point passed by G29 is the reference position used in the previous execution of G28 or G30.

Note:

If the G28 or G30 command has not been executed before the system executes the G29 instruction, the system will issue an error [0x0253 Reference position error].

Example description:

Example	Program/ Instruction
 S: Start point. B: Middle point R1: Reference point from N2.60 CP: End position	Move from reference point: Incremental command: G28G91X70.0Y40.0 M06 G29X20.0Y-40.0 Absolute command: G28G90X100.0Y80.0 M06 G29X120.0Y40.0

3.5.3 Return to 2nd, 3rd, 4th reference position (G30)

Return to 2nd, 3rd, 4th reference position		Group	Command code
Milling		00	G30
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G30	P2	CP
	P3	
	P4	

CP Intermediate position of instruction

P2 2nd reference position selection

P3 3rd reference position selection

P4 4th reference position selection

Instruction description:

The G30 command will use the G00 rapid positioning method to return to the 2nd, 3rd or 4th reference point through the intermediate point set by the command. The CP in the command format is expressed as the intermediate position. The axis whose intermediate coordinate is not specified will directly return to the corresponding reference point. The P2, P3, and P4 are the basis for the command to select the 2nd, 3rd or 4th reference point for the target position. If users want to select the 2nd reference point, the P2 can be omitted.

When tool G41 or G42 radius compensation is set, it is recommended to cancel the tool radius compensation function first. If the tool radius compensation function is not canceled, the system will move to the middle point and return to the corresponding reference point using a path without tool radius compensation when executing the G30 action. Then, when the next motion block is activated, the system will automatically restore the tool radius compensation function. In addition, the tool length compensation function of G43 or G44 will still be valid before reaching the intermediate point, and then it will return to the corresponding reference point without tool compensation when G23 is executed. The tool length compensation will be canceled and will not automatically recover anymore, users need to give the tool length compensation command again.

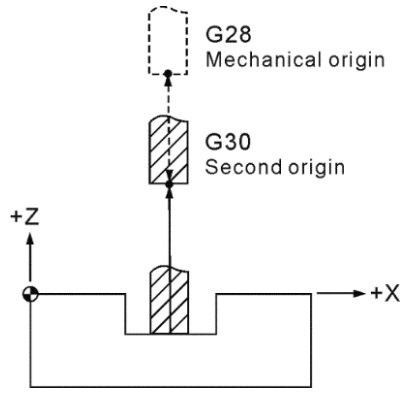
Note:

The 2nd, 3rd, 4th reference position settings are in the following parameters:

- **[N2.061 2nd reference coordinate].**
- **[N2.062 3rd reference coordinate].**
- **[N2.063 4th reference coordinate].**

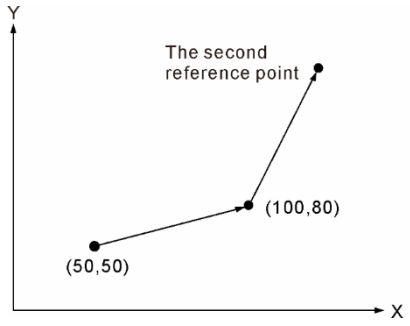
3

Example description:

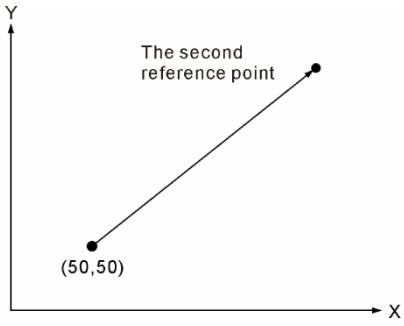
Example	Program/ Instruction
	2nd reference point return: After executing the G90 G30 P2 Z0 command, the Z axis will first move to the intermediate point position Z0, and finally move to the 2nd reference position, which completes the command action of the 2nd reference point return.

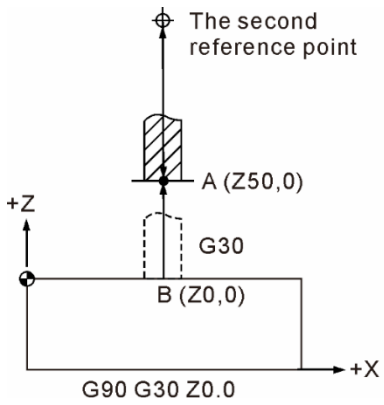
Example diagram:

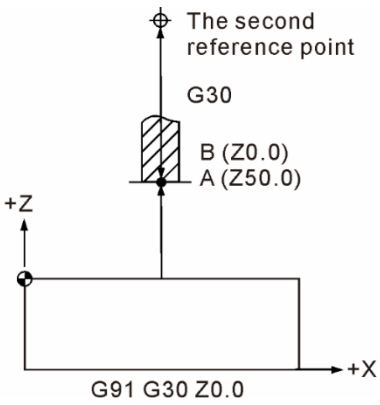
G90 G30 P2 X100. Y80.

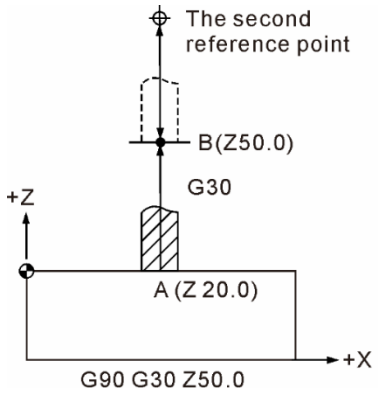


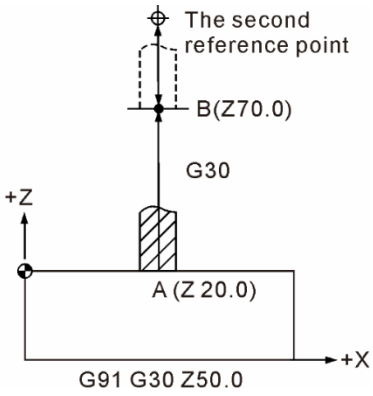
G91 G30 P2 X0. Y0. (Without intermediate point)











3.5.4 Skip function (G31)

Skip function		Group	Command code
Milling		00	G31
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G31	CP	F_	P_

CP The target position in the absolute mode.
The movement distance in the increment mode.

F_ Feed speed, the feed speed here is only available in this instruction.
It will refer to parameter **[N1.027 Skip function default speed]** if there is no command in the instruction.

P_ Set P0 or P255 or omitting the P argument. The function will be triggered by any HSI signal.
Set P1-P255, select the trigger signal using binary bits. Shown in the table below:

HSI PORT	8	7	6	5	4	3	2	1	Remark
P1	0	0	0	0	0	0	0	1	Enable HSI 1
P2	0	0	0	0	0	0	1	0	Enable HSI 2
P3	0	0	0	0	0	0	1	1	Enable HSI 1 and 2
					~				
P255	1	1	1	1	1	1	1	1	Enable HIS 1 to 8

Instruction description:

This command will move linearly to the specified position by CP, and continuously monitor the input signal status of HSI during the movement. When the signal is touched, the block command will immediately stop and then start the next block; if the specified target position is reached and the HSI signal has not been triggered yet, the system will still execute the next single block. When the skip condition is satisfied, the machine coordinates and absolute coordinate information will be written to the temporary memory right after the HSI signal is triggered and update the results into the variables listed in the following table:

Name	X	Y	Z	…	16
G31 machine coordinate record	#21048	#21049	#21050		#21063
G31 absolute coordinate record	#21064	#21065	#21066		#21079

3

Note:

(1) The skip function is not permitted to use cut tool radius compensation (G40/G41), please disable it before using.

(2) Skip function external input type, related parameters:

- Parameter **[N5.006 HSI high speed input polarity]**.
- Parameter **[N5.007 HSI high speed input switch]**.
- Parameter **[N1.013 Bit21 The initial recorded value of # variable for G31 function]**.

(3) Motion speed

- Refers to parameter **[N1.027 Skip function default speed]** when speed F is not specified.

- In the G01, the limit speed is set based on the parameter **[N1.33Cutting command maximum feed]**.

- In the G00, the limit speed is set based on the parameter **[N2.20 Axis G00 rapid command maximum speed]**.

(4) The acceleration and deceleration are used when the skip function ends without being triggered.

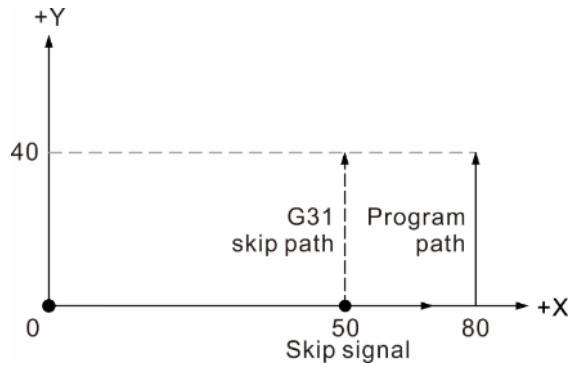
$$\text{Acc./Dec.} = \frac{\text{N1.033[Cutting command maximum feed]}}{\text{N1.028[Skip function acc. and dec. time]}}$$

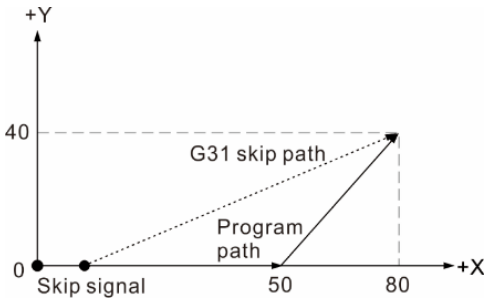
(5) Skip deceleration when skip function triggered.

$$\text{Skip deceleration} = \max\left\{\frac{\text{N1.033[Cutting command maximum feed]}}{\text{N1.028[Skip function acc.and dec.time]}} \middle| \frac{\text{N2.020[Axis G00 rapid command maximum speed]}}{\text{N2.021[Axis G00 rapid command acc.and dec.time]}}\right\}$$

(6) For the speed smoothing parameter, refer to **[N1.29 Skip function S curve time]**.

Example description:

Example	Program/ instruction
	G90 G00 X0 Y0 G01 G31 X80.0 F500.0 Y40.0 Take the above program for example. The system did not receive a skip signal while moving from position 0.0 to target 80.0. After X reaches the target, the next single block Y40.0 is executed, as shown by the solid arrow in the figure. The system stops the G31 block immediately right after it receives the skip signal before reaching the target X80.0 and then starts the next single block, as shown by the dashed arrow in the figure.



G90 G00 X0 Y0
G01 G90 G31 X50.0 F150.0
X80.0 Y40.0

Take the above program for example.

The system did not receive a skip signal while moving from position 0.0 to target 50.0. After X reaches the target, the next single block X80.0 Y40.0 is executed, as shown by the solid arrow in the figure.

The system stops the G31 block immediately right after it receives the skip signal before reaching the target X50.0 and then starts the next single block, as shown by the dashed arrow in the figure.

3.6 Coordinate system

3

3.6.1 Plane selection (G17/G18/G19)

X-Y plane selection		Group	Command code
Milling		02	G17
Lathe	A-Type		
	B-Type		
	C-Type		

Z-X plane selection		Group	Command code
Milling		02	G18
Lathe	A-Type		
	B-Type		
	C-Type		

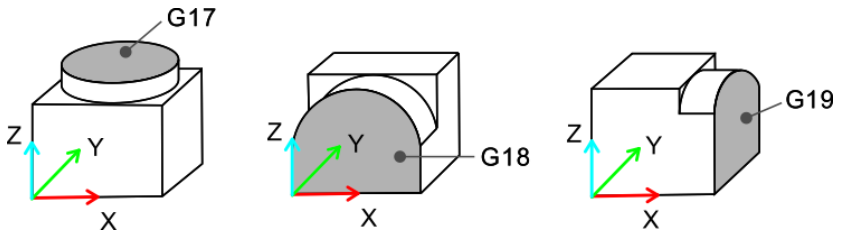
Y-Z plane selection		Group	Command code
Milling		02	G19
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G17	G01
G18	G02
G19	G03

Instruction description:

The plane selection function is used when switching between different reference planes. If the three axes are moving synchronously, there will be no need to set it. The G17 – G19 instructions are used to set the planes where arc cutting, cycle instructions or tool compensation commands are being performed.



Notes:

The system default plane is set by the parameter **[N1.20 Bit 3~4 System initial working plane]**.

3.6.2 Local coordinate system (G52)

Local coordinate system		Group	Command code
Milling		00	G52
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G52	CP

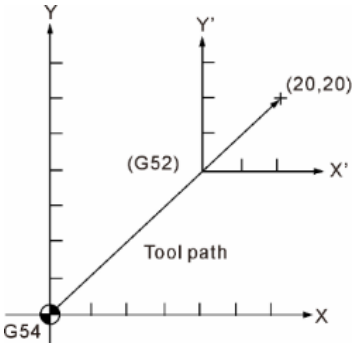
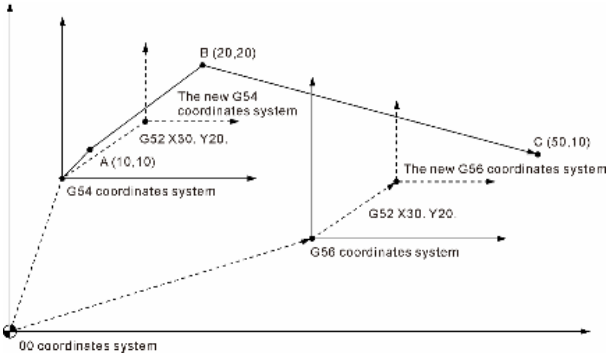
CP Local coordinate original point.

Instruction description:

In order to easily locate the coordinate position of the processing path, an offset coordinate system (also known as a sub-coordinate system) can be defined on each working coordinate system through the local coordinate system. The command method is G52 in conjunction with each axis's absolute position, and the local coordinate system can be established on the working coordinate system such as G54-G59 and G54P1-G54P64. The local coordinate system is only valid in G90 absolute mode and cannot be used in G91 incremental mode.

When canceling the local coordinate system specification, users need to specify the X, Y, and Z coordinate values in the G52 command as zero. That is to say, after executing G52 X0 Y0 Z0, the local coordinate system settings are canceled.

Example description:

Example	Program/ Instruction
Example 1: 	(Absolute coordinate) G90 G54 X0 Y0 (X00.,Y00.) G52 X40.0 Y40.0 (X40.,Y40.) G00 X20.0 Y20.0 (X60.,Y60.)
Example 2: 	G90G54G00X10.Y10. G52X30.Y20. G00X20.Y20. (A point to B point) G56G00X50.Y10. (B point to C point)

3

3.6.3 Machine coordinate system (G53)

Machine coordinate system		Group	Command code
Milling		00	G53
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G53	CP

CP Target position mechanical coordinate location point.

Instruction description:

Positioning the tool at the specified machine coordinates at rapid speed. It is generally used in applications where the reference point is established on the machine coordinates.

Notes:

(1) G53 can only be used when the system origin coordinates have been established.

(2) The G53 instruction must be programmed in the G90 absolute command, otherwise the G53 block will be ignored and not executed, but the status instructions, such as G00/G01 or G90/G91, in the block will still change the status and will affect the motion command of the following blocks.

(3) G53 is a non-continuously valid command and will only be valid in one single block.

(4) When executing the G53 command, the tool radius compensation and tool length compensation will be automatically canceled. In addition, the tool radius compensation will be automatically restored when the next motion block is executed, while the command for the tool length compensation must be given again to activate it.

(5) When there is a specified motion command in the G53 block, the axis will move to the specified point. Otherwise, the tool will stand still.

(6) The G53 and G28 cannot be programed in the same block, otherwise the system will return [0x0265 G53 and G28/30 instructions cannot be programed together].

Example description:

Program / Instruction	
G91 G53 X150. Y-150.	(This G53 instruction will be omitted)
X-30. Y-30.	(This block switches the system into incremental command mode)
G90 G53 X50. Y-50. Z0.	(Move to machine coordinate position of X50.Y-50. Z0.)
G1 G53 X100. Y-100. F1000	(Command in G01 instruction)
X50. Y50.	(This motion command is applying G01 F1000. movement)

3.6.4 Work coordinate system (G54/G55/G56/G57/G58/G59)

Work coordinate system		Group	Command code
Milling		12	G54/G55/G56/G57/G58/G59
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G54
G55
G56
G57
G58
G59

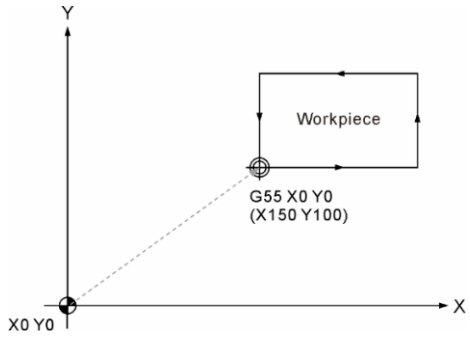
Instruction description:

Instruction description:

The working coordinate system can make it easier to perform the programing and establish multiple coordinate systems on the work platform at the same time for multiple process switching. When the working position changes, the processing path does not need to be programmed again.

Only the workpiece coordinate reference point needs to be changed to restart the processing. A simpler method of establishing a working coordinate system is to move the tool to the origin coordinates of the new coordinate system, and then register its machine coordinates in the controller's working coordinate system (G54-G59) parameters. Then the corresponding working coordinate system code can be executed to switch to a different workpiece plane.

Example description:

Example	Program/ instruction
	(G55 reference point set X-axis150.Y-axsi100.) O1000 G00 G90 G55 X0 Y0 (move to G55 reference position) G00 X100.0 (Rectangular start point) Y50.0 X0. X0. (Rectangle end point)

3

3.6.5 Extended working coordinate system (G54P_)

Extended working coordinate system		Group	Command code
Milling		12	G54P_
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G54	P_

P_ Extended coordinate ID, range 1-64.

Instruction description:

Except of standard working coordinate systems such as G54~G59, the G54 provides 64 sets of expanded working coordinate systems. The usage method is the same as the working coordinate system, please refer to section 3.6.4.

3.6.6 Coordinate system setting (G92/G50)

Coordinate system setting		Group	Command code
Milling		00	G92
Lathe	A-Type		G50
	B-Type		G92
	C-Type		G92

Instruction format:

G92	CP

CP Replaces the current system coordinate system.

Instruction description:

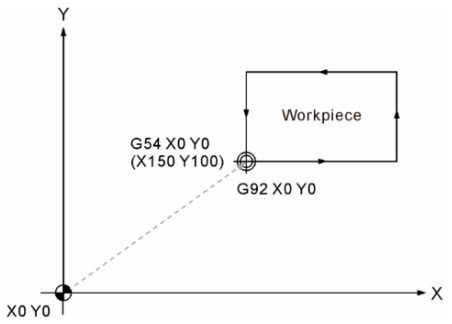
The coordinate system setting can set the current position as the new reference origin. When the CP value is zero, it means that the current tool position is set as the zero point of the new reference coordinate. The absolute instructions in this program will refer to the position based on this origin point.

Note:

(1) After the G92 command is enabled, it will be maintained until the program ends command M02/M30, or a new G92 command is issued, and the status will not be canceled or updated.

(2) After executing system RESET, the setting state of G92 will be canceled.

Example description:

Example	Program/ instruction
	(G54 reference point X150. Y100.) G00 G90 G54 X0.0 Y0.0 G00 G92 X0.0 Y0.0 (Set as program origin point) G01 X100.0 Y0.0 F1000.0 (Rectangle start point) X100.0 Y50.0 X0.0 Y50.0 X0.0 Y0.0 M30 (Program finished)

3.7 Input unit

3

3.7.1 Absolute command (G90)

Absolute command		Group	Command code
Milling		03	G90
Lathe	A-Type		-
	B-Type		G90
	C-Type		G90

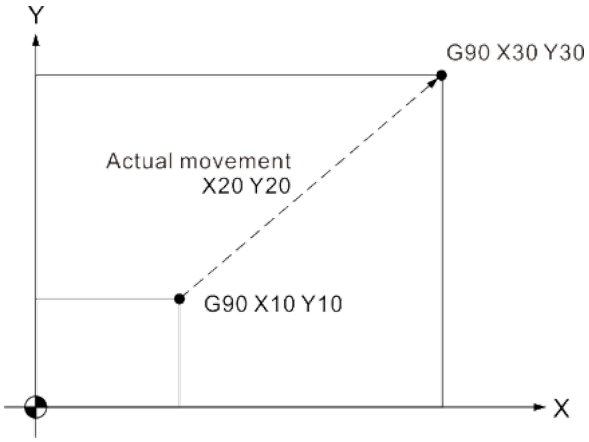
Instruction format:

G90

Instruction description:

This command is a continuously valid status command. After executing this command, the axes commands, coordinate angles, etc. will all be executed in absolute position mode. In other words, the movement of the tool is based on the working coordinate origin as the reference point, and all position commands are absolute coordinates relative to this reference point.

Example description:

Example	Program/ instruction
	O1000 G01 G90 X10.0 Y10.0 F500.0 (Absolute coordinate X10.Y10.) X30.0 Y30.0 (Absolute coordinate X30.Y30.)

3.7.2 Increment command (G91)

Increment command		Group	Command code
Milling		03	G91
Lathe	A-Type		-
	B-Type		G91
	C-Type		G91

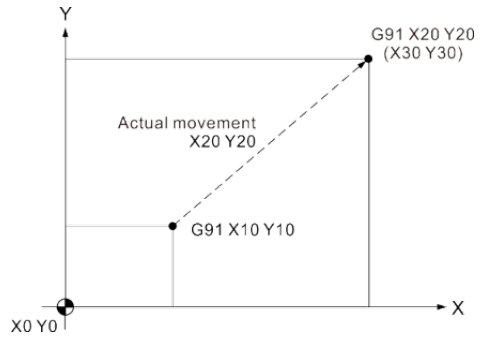
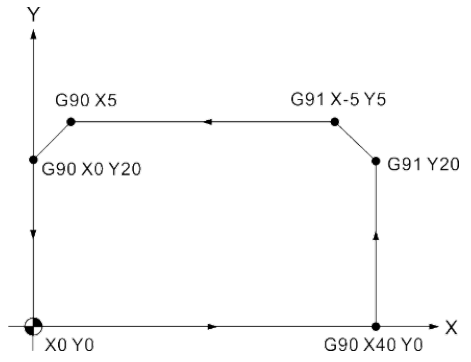
Instruction format:

G91

Instruction description:

This command is a continuously valid status command. After executing this command, the axes commands, coordinate angles, etc. will all be executed in incremental position mode. The axes movement or rotation command of a motion block is an incremental distance from its current position to the target position. G91 and G90 are mutually exclusive state instructions. When G91 is specified, the G90 state will be replaced by the G91 state.

Example description:

Example	Program/ instruction
	Relative instruction example: O1001 G01 G91 X10.0 Y10.0 F500.0 (Move X10, Y10) X20.0 Y20.0 (Move X20, Y20) When starting from X0.Y0 and the first position is X10.Y10, the second position is X20.Y20. However, the actual machine position stops at X30.Y30, as shown in the figure.
	G90 and G91 program example: O0010 G01 G90 X0 Y0 F1000 X40.0 G91 Y20.0 X-5.0 Y5.0 G90 X5.0 X0 Y20.0 Y0 M30

3

3.7.3 mm or inch units (G20/G21/G70/G71)

inch input		Group	Command code
Milling		03	G20
Lathe	A-Type		G20
	B-Type		G20
	C-Type		G70

mm input		Group	Command code
Milling		03	G21
Lathe	A-Type		G21
	B-Type		G21
	C-Type		G71

Instruction format:

G20
G21

Instruction description:

This is a command for switching between inch and mm units. This command is only valid for the linear axis movement, and the angle of the rotation axis is not affected. The system units that will be affected are as follows:

- The F value for cutting feed speed.
- Coordinate position instruction value.
- Working coordinate position.
- Tool compensation and movement value.

Note:

(1) Switching between mm and inch is prohibited during the program execution, so it must be defined at the beginning of the program.

(2) It is prohibited to use both mm and inch unit commands in the same program.

3.8 Auxiliary function

3.8.1 M-code

The main function of the M code in the NC program is to turn on or off the machine-related auxiliary functions. Except for M00, M01, M02, M30, M98 and M99 in the system, which have special function definitions, the rest can be defined by the equipment manufacturer. Please refer to the relevant instructions from the equipment manufacturer.

3.8.1.1 Program stop (M00)

When the program runs to the M00 block, the system will hold execution. When the program is placed on hold through M00, the corresponding M3x037 in the MLC will be automatically triggered, and the hold status will not be released until M3x037 is released.

Name	Property	Path 1	Path 2	Path 3	Path 4
M00 hold status	R	M31037	M32037	M33037	M34037

3.8.1.2 Optional stop (M01)

When the program executes the M01 block and the optional stop control bit of M2x009 is enabled, the system will hold execution. The optional stop status of M3x038 will be triggered when the optional stop is activated until the hold status is released.

Name	Property	Path 1	Path 2	Path 3	Path 4
Optional stop enable	R/W	M21009	M22009	M23009	M24009
Optional stop status	R	M31037	M32037	M33037	M34037

3.8.1.3 Program end (M02)

When the program executes the M02 block, it will stop running the NC program. The final execution line index will remain in the M02 block and will not return to the beginning of the program. It can be used in detect unexpected stop states because it remains in the last execution state.

3.8.1.4 Main program end M30

When the program executes the M30 block, it will stop running the NC program. The program execution line index will return to the beginning of the main program, which can be used when the program is completely executed.

3.8.1.5 Macro call (M98)

Calls sub-programs, for the functions please refer to section **[3.11.4.7 Sub-program (M98/M99)]**.

3.8.1.6 Program return (M99)

Program return, for the functions please refer to section **[3.11.4.7 Sub-program (M98/M99)]**.

3.8.1.7 Interrupt macro call (M96/97)

Calls the interrupt macro, for the functions please refer to section **[3.11.4.6 Interrupt macro call (M96/M97)]**.

3.8.2 Parameter group change (G05)

3

After the program is executed, the user can switch the processing parameters according to requirements.

Parameter group change		Group	Command code
Milling		00	G05
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G05	P_

P_ parameter group ID, range: P1-P20.

Instruction description:

After the function is executed, the system will immediately apply the selected parameter group settings, allowing users to have different processing characteristics or effects for specific processing areas. A total of 20 parameter groups are available.

G05P_: 1-20 for the group number.

Use G05 P0 to cancel the parameter group settings and return to the original processing parameter settings.

Supported parameters are as follows:

para	Channel para name
N1.030	G00 rapid command maximum speed
N1.031	G00 rapid command acc. and dec. time
N1.032	G00 rapid command S curve time
N1.033	Cutting command maximum feed
N1.034	Cutting command acc. and dec. time
N1.035	Cutting command S curve time
N1.036	Look-ahead filter time
N1.037	Look-ahead S curve time
N1.038	Arc command reference feed
N1.039	Arc command minimum feed
N1.041	Turn on speed
N1.050	Max command distance of curve fitting
N1.051	Min command degree of curve fitting
N1.052	Minimum command length of turning
N1.053	Path smoothing deviation
N1.054	Path smoothing command error

para	Axis para name
N1.096	Blending – distance before next block
N1.097	Blending – velocity %between rapid command
N1.098	Blending – velocity %between rapid and cutting command
N1.099	Blending – velocity %between cutting command
N2.020	Axis G00 rapid command maximum speed
N2.021	Axis G00 rapid command acc. and dec. time
N2.022	Axis G00 rapid command S curve time
N2.023	Axis cutting command maximum speed
N2.024	Axis cutting command acc. and dec. time
N2.025	Axis cutting command S curve time
N2.026	Axis maximum turn speed tolerance
N2.027	Blending – axis target reach distance
N2.029	Axis position filter time

3.8.3 Programmable data input (G10)

Programmable data input		Group	Command code
Milling		00	G10
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction description:

During program execution, users can update the system configuration through G10 with the L command function. Supported functions are as follows:

- L02: Register working coordinate data.
- L10: Register tool length.
- L11: Register tool length wear consumption.
- L12: Register tool radius.
- L13: Register tool radius wear consumption.
- L14: Register tool length tolerance.
- L15: Register tool actual lifetime.
- L20: Register extend working coordinate.
- L21: Register software limit position.

Note:

(1) G10 L2/10/11/12/13/14/15/20/21 commands are available for both absolute and relative command modes, and they support G code 03 group G90/G91 mode switching and U, V, W relative command input of type-A lathe machines.

(2) The G10 command is a non-continuously valid command and will only be valid in one single block. The offset coordinates and the compensation of the working coordinate system are based on the machine coordinate system.

(3) After using L2 (register working coordinate) or L20 (register extended working coordinate) to change the coordinate data, it will take effect immediately in the next motion block. To change L10-L13 (register tool compensation data), the compensation command (G41/G42 or G43/G44) and the compensation data number (D or H number) must be executed again to take effect.

Example description:

Example program	Explanation
G10 L10 P1 R-300.0	Set the 1st tool length compensation value H-300.0
G10 L10 P2 R-100.0	Set the 2nd tool length compensation value H-100.0
G90 G10 L10 P1 R-300.0 G91 G10 L10 P1 R-100.0	Set the 1st tool length as -300.0 Increase the 1st tool length -100 to -400

3.8.3.1 Register working coordinate (L02)

G10	L02	P_	CP

L02 Register working coordinate data.

P_ Parameter group ID.

ID	Coordinate selection
P0	Offset coordinate
P1	G54
P2	G55
P3	G56
P4	G57
P5	G58
P6	G59

CP Coordinate original point position.

Instruction description:

Modify the coordinate system reference point. If it is necessary to modify the extended working coordinate system, please use the L20 group function.

3.8.3.2 Register tool length (L10/11)

G10	L10	P_	R_
	L11		CP

L_ Code function.

ID	Function selection
L10	Tool length log
L11	Tool length wear consumption

P_ Parameter group ID.

ID	Tool selection
P0	Invalid ID
P1-P512	Corresponding tool ID 1-512

R_ Lathe: Tool radius compensation data.

Other: Z-axis length/ wear consumption.

CP Tool length/ wear consumption.

Q_ Lathe: Tool tip direction data of the hypothetical tool.

Other: Not support.

Instruction description:

G10 L10–L12 sets the tool length compensation value.

P is the tool number index.

When the machine type is not lathe and then L10/11 contains the tool information for both the R_ and CP command, the setting of CP will be ignored.

3.8.3.3 Register tool radius (L12/13)

G10	L12 L13	P_	R_

L_ Code function.

ID	Function selection
L12	Tool radius
L13	Tool radius wear consumption

P_ Parameter group ID.

ID	Tool selection
P0	Invalid ID
P1-P512	Corresponding tool ID 1-512

R_: Tool radius data.

Instruction description:

G10 L12-L13 are instructions only for milling applications, which set the compensation value of the tool radius.

P is the tool number index.

R represents tool radius registration data when it is at L12 and represents tool radius wear consumption when it is at L13.

3.8.3.4 Register tool tolerance (L14)

G10	L14	P_	R_ CP

L14 Register tool length tolerance.

P_ Parameter group ID.

ID	Tool selection
P0	Invalid ID
P1-P512	Corresponding ID 1–512 tool data

R_ Z axis length tolerance.

CP Tool length tolerance.

Instruction description:

G10 L14 sets the tool length tolerance value.

P is the tool number index. When L14 contains both the R_ and CP command, the setting of CP will be ignored.

3.8.3.5 Register tool actual lifetime (L15)

G10	L15	P_	D_	R_
				CP

L15 Register tool actual lifetimes.

P_ Parameter group ID.

ID	Tool selection
P0	Invalid ID
P1-P512	Corresponding ID 1 - 512 tool data

D_ Mode ID.

ID	Tool selection
D0	Invalid ID
D1	During Z-axis execution, increase the target lifetime incrementally

R_ Actual lifetimes of Z axis.

CP Tool lifetime.

Instruction description:

G10 L15 sets the tool lifetime value.

P is the tool number index.

When L14 contains both the R_ and CP command, the setting of CP will be ignored.

3.8.3.6 Register extend working coordinate (L20)

G10	L20	P_	CP

L20 Register extend working coordinate.

P_ Parameter group ID.

ID	Coordinate selection
P0	Invalid parameter
P1	G54P1
P2	G54P2
~	
P64	G54P64

CP Coordinate original point position.

Instruction description:

Modify the extend coordinate reference point.

3.8.3.7 Register software limit position (L21)

G10	L21	P_	CP

L20: Register extend working coordinate.

P_: Parameter group ID.

ID	Limit selection
P0	Invalid parameters
P1	Set 1st positive software limit
P2	Set 1st negative software limit
P3	Set 2nd positive software limit
P4	Set 2nd negative software limit

CP: Software limit position.

Instruction description:

Modifies the system software limit position.

3.8.4 Dwell (G04)

Dwell		Group	Command code
Milling		00	G04
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G04	P_
	X_

X_ pause time, unit: sec. range: 0.001-99999.999.

P_ pause time, unit: ms, range: 1-99999999.

Instruction description:

This instruction is a command for the system to pause the time.

The X argument sets the pause time in seconds, decimal points are allowed.

The P argument sets the pause time in milliseconds (0.01 seconds), decimal points are not allowed in this format.

Note:

The X argument is affected by the **[N1.11 bit4 Command block ignore decimal point]**.

When the setting is 0 to disable decimal point ignoring, if the command is [G04 X1], the pause time will be 0.001 seconds, and if the command is [G04 X1.], the pause time will be 1 second.

When the setting is 1 to enable decimal point ignoring, if the command is G04 X1, the pause time will be 1 second.

Example:

Example program	Explanation
G04 X1.5	Block pause 1.5 sec
G04 P1500	Block pause 1500 ms

3

3.8.5 MLC M relay control (G900)

MLC M relay control		Group	Command code
Milling		00	G900
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G900	P_	Q_

P_ M relay number. Range: 30200~30339.

Q_ Control value. Range: -1.

Instruction description:

This instruction controls the status of the MLC M relay through a single block instruction.

P_ Specifies the MLC M relay number.

Q_ Specifies the status to be updated. When the value is 0, it writes OFF, and when it is not 0, it writes ON.

Notes:

(1) When the M number specified by P_ exceeds the range, the block is automatically ignored and not executed.

(2) When P_ or Q_ is not specified, this block is automatically ignored and not executed.

(3) When Q_ uses an out-of-range value, it will automatically be executed with a value of 1.

Example:

Program example	Description
G900 P30200 Q1	Set the M30200 in MLC as ON
G900 P30320 Q0	Set the M30200 in MLC as OFF

3.9 Spindle function

3.9.1 S-code instruction

S code is used for specified spindle rotation, format is S□□□□□□P□.

S specified spindle speed, range: 0-600000.

P specified spindle number, range 1~4.

The 1st spindle will be selected when the command is not specified.

Related macro variables:

	Variables
S_P1 command value	#20404
S_P2 command value	#20405
S_P3 command value	#20406
S_P4 command value	#20407

MLC related register:

	Status	Property	Path 1	Path 2	Path 3	Path 4
S-code execution	BIT	R	M31065	M32065	M33065	M34065
S_P1 command value	Float	R	D31026	D32026	D33026	D34026
S_P2 command value	Float	R	D31030	D32030	D33030	D34030
S_P3 command value	Float	R	D31364	D32364	D33364	D34364
S_P4 command value	Float	R	D31376	D32376	D33376	D34376

Note:

(1) The S code function will take effect only when the spindle in the path is enabled.

(2) The setting parameters for the maximum speed limit of each spindle are N1.008, N1.058, N1.108, and N1.158 **[Maximum spindle speed]**.

(3) When the spindle speed is controlled by the MLC, please refer to the machine manufacturer's instructions for related control functions.

3

3.9.2 Spindle linear speed control (G96/G97)

Constant linear speed control		Group	Command code
Milling		17	G96
Lathe	A-Type		
	B-Type		
	C-Type		

Constant linear speed control cancel		Group	Command code
Milling		17	G97
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G96 / G97	S_

G96S_ Specified spindle linear speed control per min, unit: m/min.

G97S_ Specified spindle number of turns per min, unit: RPM.

Instruction description:

G96 specifies that the spindle is controlled at a linear speed per minute (m/min). Taking G96 S200 as an example, the tool will run at a cutting speed of 200 meters per minute. When the lathe cutting speed is fixed, the spindle speed will vary depending on the diameter of the material in the X direction. When the processing diameter is smaller, the spindle speed will be faster, and when the processing diameter is larger, the spindle speed will be slower. The main purpose of this instruction is to maintain consistent finishing on a workpiece.

G97 specifies that the spindle is controlled in revolutions per minute (rev/min). Taking G97 S2000 as an example, the spindle speed will be maintained at 2000 revolutions per minute. This command can be used to fix the spindle speed during thread processing, drilling, and tapping.

Note:

The constant linear speed of G96 is limited by the parameters **N1.008**, **N1.058**, **N1.108** and **N1.158 [maximum spindle speed]**, and program instructions can also be used to limit the maximum spindle speed to avoid the spindle rotation speed exceeding the usage limit accidentally.

Max spindle speed clamp		Group	Command code
Milling		00	G92 S_
Lathe	A-Type		G50 S_
	B-Type		G92 S_
	C-Type		G92 S_

Example description：

Program description	
G92 S200	(Apply G92 to limit the spindle maximum speed to 200RPM)
G96 S130 M03	(Cutting speed remains at 130 m/min)
…….	
G97 S1300 M03	(Represents the spindle speed remaining at 1300 RPM)
…..	

3.10 Tool function

T code instructions are used to control machine tools. When the system executes the T code in the NC program, it will pass the T code argument to the MLC or the O9000 macro program for the logic program to judge the tool exchange action. For the tool exchange process, please refer to the equipment manufacturer manual.

3.10.1 Lathe tool

The T instruction includes a 4-digit code in the lathe system, the format is T□□□□, composed of the tool number and tool compensation code.

Notes:

(1) The tool compensation number can be cleared through system Reset, M30/M02, or G28/G30, but the tool number will still be retained.

(2) To read the T code information on the HMI, the settings are as follows:

Read memory location: G_1_T_0

Numerical value: Word,

Numerical format: Unsigned Decimal

Integer number: 4

(3) When the parameter **[N1.10 Bit22 T code call macro O9000]** is enabled, the T code value will not be displayed.

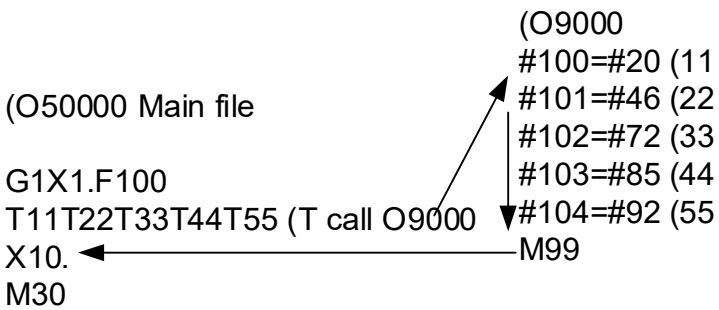
3.10.2 Milling multiple tools

To use the function of multiple tools on a single block, the parameter **[N1.10 Bit22 T code call macro O9000]** must be enabled, and up to five T codes are allowed to be input in a single block. The #20, #46, #72, #85, and #92 in the O9000 is for updating the arguments of the T code.

T code	O9000 local variable
1st group of T code	#20
2nd group of T code	#46
3rd group of T code	#72
4th group of T code	#85
5th group of T code	#92

T code update to # variable corresponding table

Example



3.11 Macro program

3

This section introduces how to use macro programs. The naming rule of macro files is to use O as the first character, and the number range is O00001-O9999. However, the file names O50000 and O60000 are reserved for main programs and MDI programs, so they cannot be used for general purposes.

The macro instructions are not affected by the block hold under the program preview mechanism. Even when single-block pause is enabled, the macro syntax will not stop decoding; however, the behavior can be changed through the parameter **[N1.10 Bit24 Look-ahead stop of # variable assignment command]** and parameter **[N1.10 Bit4 Enable Look-ahead of judgment command]**. The following instruction applications are considered macro commands:

(1) Program sections with the operation instruction (=).
(2) Program sections with the control instructions (GOTO, IF, WHILE).
(3) Program sections with the call macro program instructions (G65, G66, G67, G-code call macro, M-code call macro).

3.11.1 # Macro variable

The # macro variables include two categories: user variables and system variables. This section explains the user variables when programming. Please refer to appendix B for system special variable functions.

The # macro variable uses double precision floating point format (Double Float), and its data length is 8 bytes.

The HMI system screen supports the quick search function, please enter "S + variable number" in the text field.

Category	Quantity	Range	Type	No. of layers	Retentive	Path public	read	write
Null variable	1	#0	NULL	1	-	-	O	-
Local variable	99	#1 ~ #99	Double	9 Note1	X	X	O	O
Public variable	900	#100 ~ #999	Double	1	X	△ Note2	O	O
Retentive variable	2000	#2000 ~ #3999	Double	1	O	△ Note3	O	O
Extend variable	8000	#4000 ~ #11999	Double	1	O	O	O	O

O: support -: not support △: partially supported.

Note 1: The maximum number of local variable layers is 9, one layer for subroutines called by M98, and 8 layers for macro programs called by G65.

Note 2: Set the parameter **[N1.10 Bit1 Regular public # variable]** to 1 to enable multi-channel sharing.

The parameter **[N0.50 Amount of public regular variable]** sets the usage range after the shared variable #100.

Note 3: Set the parameter **[N1.10 Bit2 Retentive public # variable]** to 1 to enable multi-channel public sharing.

The parameter **[N0.51 Amount of public retentive variable]** sets the usage range after the shared variable #2000.

3

3.11.2 Operation instruction

3.11.2.1 Note

Name	Instruction	Application and instruction
Note	(	In a single block, all text after the symbol will be regarded as comment content, and will not be included in program execution.
	//	
	%	
	;	
Program name tag	O	When the 'O' is the 1st character of the program, the 1st line will be the file description and will not be included in program execution.

3.11.2.2 Value calculation

Numerical operations can be performed to perform calculations such as four-rule operations, trigonometric functions, etc. on # variables. The following are instruction descriptions:

Name	Instruction	Application and instruction
Negative	-**a**	The negative value of **a**.
Addition	**a** = **b** + **c**	**a** is the sum of **b** plus **c**. For example: #100=1+2, the result of #100 will be 3.
Subtraction	**a** = **b** - **c**	**a** is the difference derived from **b** minus **c**. For example: #100=1-2, the result of #100 will be -1.
Multiplication	**a** = **b** * **c**	**a** is the product of **b** times **c**. For example: #100=2*3, the result of #100 will be 6.
Division	**a** = **b** / **c**	**a** is the quotient of **b** divided by **c**. For example: #100=2/3, the result of #100 will be 0.667.
Remainder	**a** = **b** MOD **c**	**a** is the remainder after dividing **b** by **c**, the decimals will be rounded off automatically. For example: #100=5MOD3, the result of #100 will be 2.
Sine	**a** = SIN [**b**]	**a** is the result of the SIN [**b**] calculation.
Cosine	**a** = COS [**b**]	**a** is the result of the COS [**b**] calculation.
Tangent	**a** = TAN [**b**]	**a** is the result of the TAN [**b**] calculation.
Arcsine	**a** = ASIN [**b**]	**a** is the result of the ASIN [**b**] calculation. When the value range of **b** exceeds -1–1, it will result in the alarm [0x257: numerical value error].
Arccosine	**a** = ACOS [**b**]	**a** is the result of the ACOS [**b**] calculation. When the value range of **b** exceeds -1–1, it will result in the alarm [0x257: numerical value error].
Arctangent	**a** = ATAN [**b**]	**a** is the result of the ATAN [**b**] calculation.
Arctangent	**a** = ATAN [**b**,**c**]	**a** is the result of the ATAN [**b**,**c**] calculation.
Square root	**a** = SQRT [**b**]	**a** is the result of the square root of the **b** value. For example: #100= SQRT [4.], the result of #100 will be 2.0.

Name	Instruction	Application and instruction
Absolute	**a** = ABS [**b**]	**a** is the result after taking the absolute value of **b**. For example: #100= ABS [-2.5], the result of #100 will be 2.5.
Rounding	**a** = ROUND [**b**]	**a** is the calculation result of the **b** value after rounding off the decimal point. When b is a negative number, the calculated content has the same absolute value as the content calculated as a positive number. For example: #100= ROUND [2.5], the result of #100 will be 3.0. For example: #101= ROUND [-2.5], the result of #101 will be -3.0.
Unconditional round down	**a** = FIX [**b**]	**a** is the result of **b** value after unconditionally rounding off the decimal point. For example: #100= FIX [2.5], the result of #100 will be 2.0. For example: #101= FIX [-2.5], the result of #101 will be -2.0.
Unconditional round up	**a** = FUP [**b**]	**a** is the result of unconditionally rounding up the decimal point of the value b. For example: #100=FUP [2.5], the result of #100 will be 3.0. For example: #101=FUP [-2.5], the result of #101 will be -3.0.
Logarithm	**a** = LN [**b**]	**a** is the calculation result of taking the natural logarithm of the value **b**. When the value of **b** is smaller than 0, it will result in the alarm [0x257: numerical value error]. For example: #100=LN [86.], the result of #100 will be 4.454.
Exponent	**a** = EXP [**b**]	**a** is the calculation result after an exponent to the **b** power. When the value of b is outside the system's limitations, it will result in the alarm [0x257: numerical value error]. For example: #100=EXP [2.], the result of #100 will be 7.389.
Power	**a** = POW [**b**,**c**]	**a** is the calculation result when **b** to **c** power. When the value of **b** is smaller than 0, it will result in the alarm [0x257: numerical value error]. For example: #100= POW [2.,3.], the result of #100 will be 8.0.

3.11.2.3 Binary operation

3

In the NC5 system, instructions for bit operations will be converted into 32-bit unsigned integers for calculation processing. Since the data of the # variable is a floating-point number, the calculation results will still be floating-point after being saved back to the # variable. For example: 3.56 AND 6 will be converted into (3 AND 6) for calculation.

Name	Instruction	Application and instruction
AND	**a** = **b** & **c** **a** = **b** AND **c**	**a** is the result of an "AND" operation on the integer data of **b** and **c**. For example: #100 = 3AND6, the result of #100 will be 2.0.
OR	**a** = **b** | **c** **a** = **b** OR **c**	**a** is the result of an "OR" operation on the integer data of **b** and **c**. For example: #100 = 3OR6, the result of #100 will be 7.0.
XOR	**a** = **b** XOR **c**	**a** is the result of an "XOR" operation on the integer data of **b** and **c**. For example: #100 = 3XOR6, the result of #100 will be 5.0.
Complement	**a** = COMPL [**b**]	**a** is the result of a complement operation on the integer data of **b**.
Bit left shift	**a** = BSL [**b**]	**a** is the result of shifting the b value to the left by 1 bit. For example: #100= BSL [0], the result of #100 will be 1.0 For example: #101= BSL [1], the result of #101 will be 2.0 For example: #102= BSL [2], the result of #102 will be 4.0 For example: #103= BSL [3], the result of #103 will be 8.0.
Bit obtain	**a** = BIT [**b**,**c**]	**a** is the state of the **c**-th bit in b integer data, the result is represented as 0 or 1. For example: #100=BIT [2,0], the result of #100 will be 0.0 For example: #101=BIT [2,1], the result of #101 will be 1.0

3.11.2.4 Logical operation

The results of logical operations can be saved in the # variable, with 0 representing False and 1 representing True; it can also be used as a judgment expression for process instructions, such as IF, ELSEIF, WHILE, etc.

Name	Instruction	Application and instruction
NOT	**a** = !**b**	If the value of **b** is 0, **a** will be 1 (True), otherwise it will be 0 (False). For example: #100=!0, the result of #100 will be 1. For example: #101=!1, the result of #101 will be 0. For example: IF [!0] GOTO2, after execution the command line will jump to N2.
AND	**a** = **b** && **c**	If the value of **b** and **c** are both 1, **a** will be 1 (True), otherwise it will be 0 (False). For example: #100=1&&0, the result of #100 will be 0. For example: #101=1&&1, the result of #101 will be 1. For example: IF [1&&0] GOTO2, after execution the command line will jump to N2.
OR	**a** = **b** || **c**	If any of the values of **b** or **c** is 1, **a** will be 1 (True), otherwise it will be 0 (False). For example: #100=1||0, the result of #100 will be 1. For example: #101=0||0, the result of #100 will be 0. For example: IF [1||0] GOTO2, after execution the command line will jump to N2.
Less than	**a** = **b** < **c** **a** = **b** LT **c**	If the value of **b** is smaller than **c**, **a** will be 1 (True), otherwise it will be 0 (False). For example: #100=9LT5, the result of #100 will be 0. For example: #101=0LT5, the result of #101 will be 1. For example: IF [1LT0] GOTO2, after execution the command line will jump to N2.
Greater than	**a** = **b** > **c** **a** = **b** GT **c**	If the value of **b** is greater than **c**, **a** will be 1 (True), otherwise it will be 0 (False).
Less than or equal	**a** = **b** <= **c** **a** = **b** LE **c**	If the value of **b** is less than or equal to **c**, **a** will be 1 (True), otherwise it will be 0 (False).
Greater than or equal	**a** = **b** >= **c** **a** = **b** GE **c**	If the value of **b** is greater than or equal to **c**, **a** will be 1 (True), otherwise it will be 0 (False).
Equal	**a** = **b** == **c** **a** = **b** EQ **c**	If the value of **b** is equal to **c**, **a** will be 1 (True), otherwise it will be 0 (False).
Not equal	**a** = **b** != **c** **a** = **b** NE **c**	If the value of **b** is not equal to **c**, **a** will be 1 (True), otherwise it will be 0 (False).

3.11.3 Process control

Process control instruction table.

3

Function	Instruction	Description
GOTO	GOTO**n**	Unconditionally jump to the label **N** line.
IF..GOTO	IF[ ]GOTO**n**	If the condition is true, then jump to the label **N** line.
IF..THEN	IF[ ]THEN	If the condition is true, then execute the program after THEN. If the condition is not true, execute the next ELSEIF, ELSE or ENDIF judgment.
	ELSEIF[ ]THEN	If the condition is true, then execute the program after THEN. If the condition is not true, execute the next ELSEIF, ELSE or ENDIF judgment.
	ELSE	ELSE is the statement when the other judgments are not true, and the program will continue to execute.
	ENDIF	End statement of IF statement. ENDIF cannot be omitted
WHILE	WHILE[ ]DOm	When the conditional is true, the loop between DO and END of layer m is executed. When the conditional is not true, the program section after END is executed. **m** is the number of loop layers, with a maximum of 3 layers.
	EXITW	Interrupts the WHILE loop; skip the current WHILE loop to END**m**.
	ENDm	Layer of the **m** loop where the program ends.

3.11.3.1 Command jump (GOTO)

Jump to specified N label position in the same program.

Instruction format:

GOTO	N

N N label number to go.

Instruction description:

(1) Jump to the line with label N mark in the same program.
If the N label does not exist, the system will issue an alarm [0x022A: N Label not found].

(2) The N label number can be described by constants as well as with #variables, such as GOTO#100.

(3) GOTO cannot be used to escape the WHILE loop.

(4) The IF [ ] THEN…ENDIF process prohibits the use of GOTO to escape.

(5) If the program contains multiple identical N label numbers, the search will start from the current line of the program to the next N label line. However, please avoid using multiple identical N tags in the same program.

Program example:

Example	Program
Example 1: N001G90G00X0.Y0. N002GOTO4 N003X100. N004Y100. N005M30	When executing to the N002 block, the system will jump to the N004 label block, so the N003 block is ignored. The target absolute coordinates finally stop at the X0.Y100. position.
Example 2: #100=5.0 N001G90G00X0.Y0. N002GOTO#100 N003X100. N004Y100. N005M30	When executing to the N002 block, the system will jump to the N005 label, which is specified by #100, so the N003 and N004 blocks are ignored. The target absolute coordinates finally stop at the X0.Y0. position.

3.11.3.2 Conditional expression (IF)

3

Determines whether the condition satisfies the statement. This function supports jump and macro program usage.

Instruction format 1:

IF [statement] GOTO N

When the condition of [statement] is satisfied, the system will jump to the block labeled N. Otherwise, the program continues to execute the next block; this format cannot be used together with THEN, ELSE, ELSEIF, and ENDIF.

Instruction format 2:

```
IF [ statement 1 ] THEN
  IF program
ELSEIF [ statement 2 ] THEN
  ELSEIF program
ELSE
  ELSE program
ENDIF
```

Instruction description:

(1) When the IF condition is satisfied, the system will continue to execute the IF program segment; otherwise, jump to the next ELSEIF, ELSE, or ENDIF instructions.

(2) When the ELSEIF condition is satisfied, the system will continue to execute the ELSEIF program segment; otherwise, jump to the next ELSEIF, ELSE, or ENDIF instructions.

(3) ELSE continues to execute the program of the ELSE program segment until ENDIF is used as the end instruction when the previous IF and ELSEIF conditions are not satisfied.

(4) IF and ENDIF are necessary syntax and cannot be omitted.

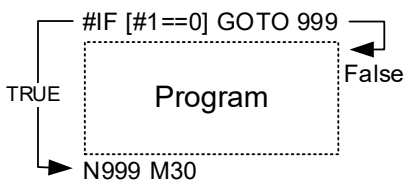
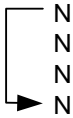
(5) ELSEIF and ELSE can be omitted depending on the requirements.

(6) If a nested structure is adopted, the IF instruction is limited to a maximum of 5 layers.

(7) Conditional expression symbol table:

Name	Symbol	Name	Symbol
NOT	!	OR	||
AND	&&	Equal to	== EQ
Less than	< LT	Less than or equal to	<= LE
Greater than	> GT	Greater than or equal to	>= GE

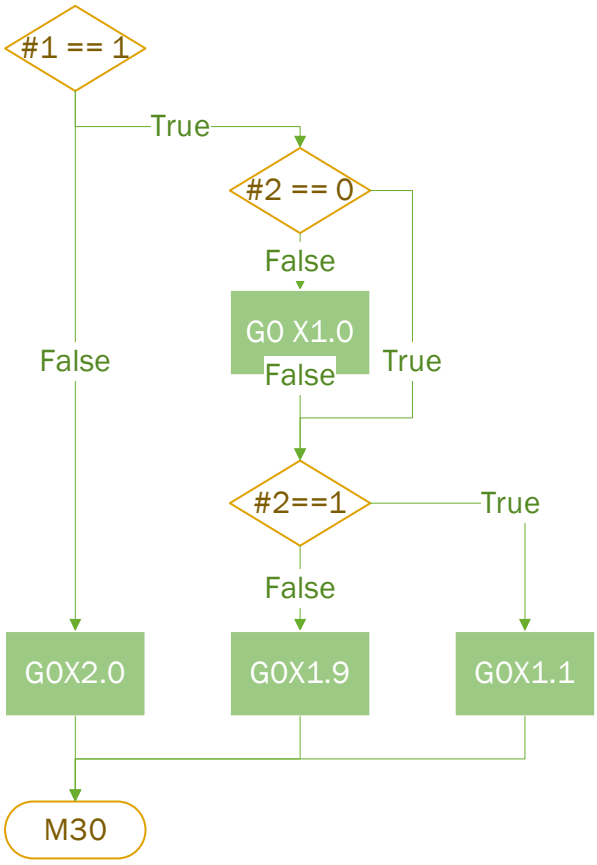
Program example:

Example	Explanation
Example 1: 	When the value of available #1 is 0, the program jumps to the N999 block, otherwise the program continues with the next block.
Example 2:  #100=5.0 N001G90G00X0.Y0. N002GOTO#100 N003X100. N004Y100. N005M30	Execute to the N002 block, the program jumps to label N5, which is specified by the #100. The N003, N004 blocks are omitted. The target absolute coordinates finally stop at the X0.Y0. position.
Example 3: IF[#1==1]THEN #100 = 0 ENDIF	If the value of #1 is 1, then the system will set #100 to 0.
Example 4: IF[[#1==#2]AND[#3==#4]]THEN #100 = 0 ENDIF	If the value of #1 is equal to #3 and #2 is equal to #4, then the system will set #100 to 0.

Example 5:
IF[#1==1] THEN
#100=1
ELSEIF[#1=2] THEN
#101=1
ELSEIF[#1=3] THEN
#102=1
ELSE
#103=1
ENDIF
	If #1 is 1, the system will set #100 to 1.
	Otherwise check if #1 is 2, then set #101 to 1.
	Otherwise check if #1 is 3, then set #102 to 1.
	If the above statements are not satisfied, then set #103 to 1.

3

Example	Explanation

Example 6:
IF[#1==1] THEN
IF[#2==0]THEN
G0X1.0
ELSEIF[#2==1] THEN
G0X1.1
ELSE
G0X1.9
ENDIF
ELSE
G0X2.
ENDIF
M30
	Example of double-layer nest structure
	If #1 is 1, check if #2 is 0, then move the X axis to 1.0; otherwise check if #2 is 1, then move the X axis to 1.1.
	If the above #2 statements are not satisfied, then move the X axis to 1.9. If #1 is not 1, then move the X axis to 2.0.
	 

3.11.3.3 Loop (WHILE)

Instruction format 1:

```
WHILE [statement] DO1
  WHIEL program
END1
```

When the [statement] is true, the [WHILE program] will be executed, and then the [statement] will be judged again after completion. If it is still true, the system will repeatedly execute the [WHILE program] and continue to repeat the above process until the [statement] is not satisfied.

Instruction description:

(1) When [statement] is true, repeatedly execute the [WHILE program].

(2) When [statement] is not true, jump to the next block after the END instruction.

(3) There is no limit on the number of blocks between DO and END.

(4) Up to 3 layers of nested loops are supported, and END1, END2, and END3 are necessary for each layer to exit the WHILE in sequence.

(5) Conditional expression symbol table:

Name	Symbol	Name	Symbol
NOT	!	OR	||
ND	&&	Equal	== EQ
Less than	< LT	Less than or equal to	<= LE
Bigger than	> GT	Bigger than or equal to	>= GE

Note:

(1) The EXITW instruction must be used to interrupt the WHILE loop. It is prohibited to use GOTO to escape from the current loop.

(2) It is prohibited to use GOTO to jump into the program segment of the WHILE loop.

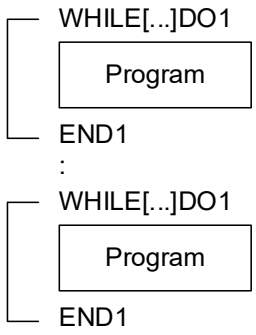
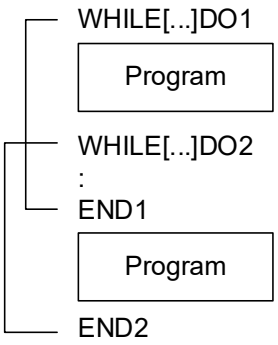
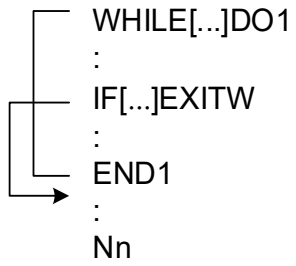
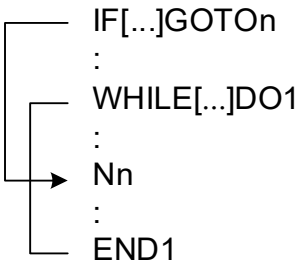
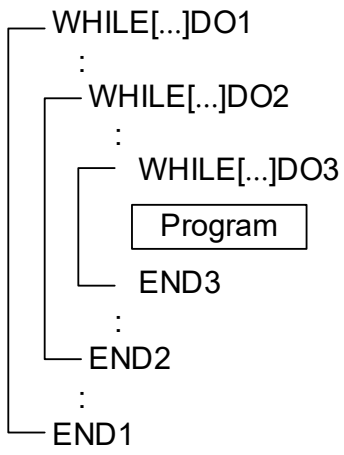
(3) The m value after DOm and ENDm is the identification code that specifies the loop layer. Since a maximum of 3 layers of nested loops are supported, only 1, 2, and 3 can be used as identification codes. The system will issue an alarm once the identification code goes beyond 1, 2, and 3.

(4) Identification codes 1, 2, and 3 can be used multiple times between DO and END.

(5) When repeated loops are intertwined with each other, the system issues an alarm, "END statement cannot be found."

3

Example description:

Correct grammar	Error grammar
Layer (1, 2, 3) can be used multiple times as required. 	The range of DO cannot overlap. 
The escape instruction can jump out of the WHILE loop to END1 to escape the loop. 	Jump instructions cannot jump from the outer layer to the loop segment. 
WHILE DO can accept a maximum of 3 layers. 	

3.11.4 Macro call

The macro calling method determines whether to support passing the values of local variables. The descriptions are as shown in the following table:

Sub-program calling mode	Type	Local variable
M98 call sub program	Sub-program	Not supported
M96 Call interrupt macro program	Macro program	Not supported
After the breakpoint search, call the macro program	Macro program	Not supported
Call initial macro before program execution	Macro program	Not supported
One-button call macro program	Macro program	Not supported
G65 Macro call program	Macro program	Supported
G66 Macro modal call program	Macro program	Supported
G code call macro program	Macro program	Supported
M code call macro program	Macro program	Supported
T code call macro program	Macro program	Supported

When using a G65, G66, G, M, T etc. macro call function that supports local variables, the argument value after the instruction will be passed into the #local variable for use in macro-operations. If arguments are not used during the call, the system will assign the corresponding variable a NULL value. Please refer to the following table for the corresponding relationships between argument codes and local variables:

(For example, A5.is expressed as the value of variable #1 is 5.0)

Code	1st argument	2nd argument	3rd argument	4th argument	5th argument
A	#1	#27	#53	-	-
B	#2	#28	#54	-	-
C	#3	#29	#55	-	-
D	#4	#30	#56	-	-
E	#5	#31	#57	-	-
F	#6	#32	#58	-	-
H	#8	#34	#60	-	-
I	#9	#35	#61	-	-
J	#10	#36	#62	-	-
K	#11	#37	#63.	-	-
M	#13	#39	#65	-	-
Q	#17	#43	#69	-	-
R	#18	#44	#70	-	-
S	#19	#45	#71	-	-
T	#20	#46	#72	#85	#92
U	#21	#47	#73	-	-
V	#22	#48	#74	-	-
W	#23	#49	#75	-	-
X	#24	#50	#76	-	-
Y	#25	#51	#77	-	-
Z	#26	#52	#78	-	-

3

3.11.4.1 Breakpoint search macro

When the parameter **[N1.020 Bit22 Call O9030 before execute break and search]** is enabled and the breakpoint search function is executed, the program will first execute the macro program of O9030 and fill in the macro variables with the program running information before the breakpoint. The relevant variables are shown in the following table:

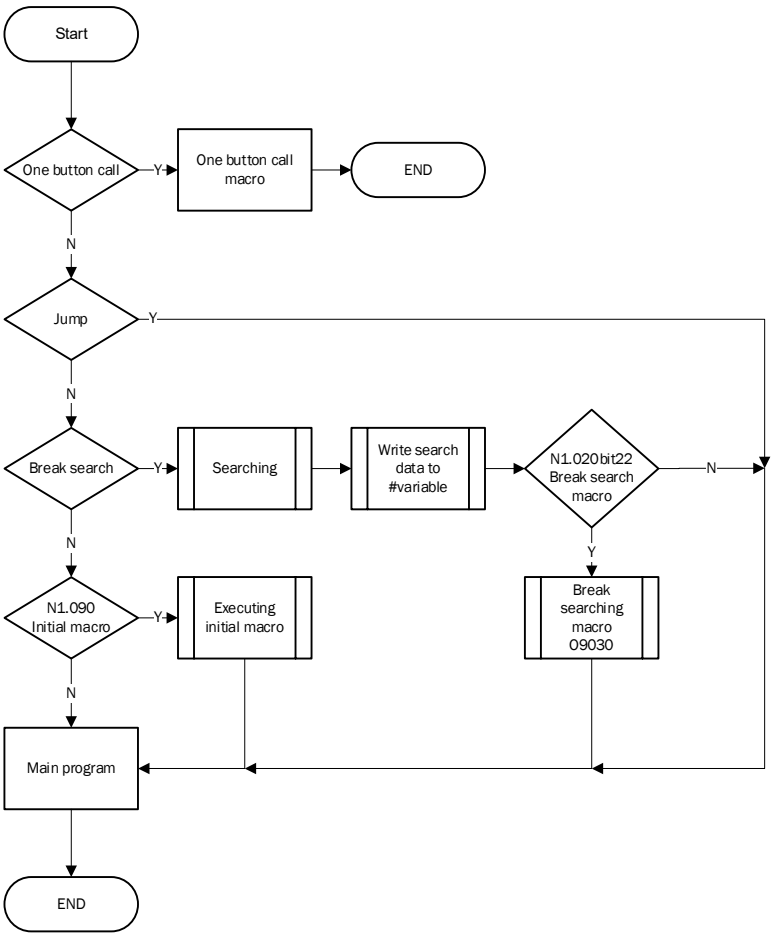
Function	Variables
M-code usage record before breakpoint search	#24042~24076
S-code usage record before breakpoint search	#24077
T-code usage record before breakpoint search	#24078~24079
The last end-point record before breakpoint search	#24080~24095
The last tool number record before breakpoint search	#24100~24103

3.11.4.2 Initial macro

Before the processing program is executed, the system will use the macro number set by the parameter **[N1.090 Initial Macro number]** as the initial macro. Therefore, when in automatic mode, the specified initialization macro program will be executed once before executing the main program.

3.11.4.3 One button call macro

One-button call is an execution signal triggered by the operator, which is then received through the MLC to execute the specified macro program. The MLC can execute macros as needed based on different conditions. Please refer to the [MLC Application Manual] for detail description.



3.11.4.4 Macro call (G65)

Macro call		Group	Command code
Milling		00	G65
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G65	P_	L_	I_

P_ Program number.

L_ Repeated times

I_ Argument value

Instruction description:

G65 is used to execute the macro program specified by the P argument, and also supports passing arguments to local variables. Please refer to the following table for the relationship between the argument code and local variable transfer:

code	1st argument	2nd argument	3rd argument	4th argument	5th argument
A	#1	#27	#53	-	-
B	#2	#28	#54	-	-
C	#3	#29	#55	-	-
D	#4	#30	#56	-	-
E	#5	#31	#57	-	-
F	#6	#32	#58	-	-
H	#8	#34	#60	-	-
I	#9	#35	#61	-	-
J	#10	#36	#62	-	-
K	#11	#37	#63.	-	-
M	#13	#39	#65	-	-
Q	#17	#43	#69	-	-
R	#18	#44	#70	-	-
S	#19	#45	#71	-	-
T	#20	#46	#72	#85	#92
U	#21	#47	#73	-	-
V	#22	#48	#74	-	-
W	#23	#49	#75	-	-
X	#24	#50	#76	-	-
Y	#25	#51	#77	-	-
Z	#26	#52	#78	-	-

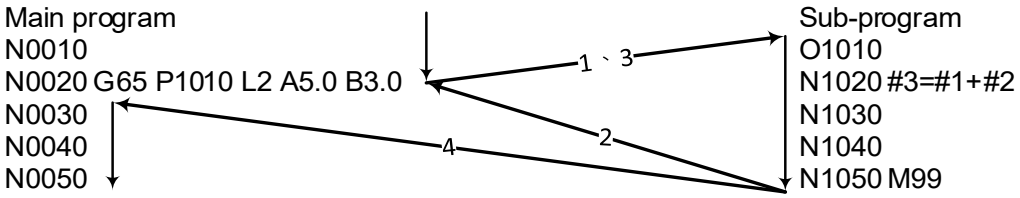
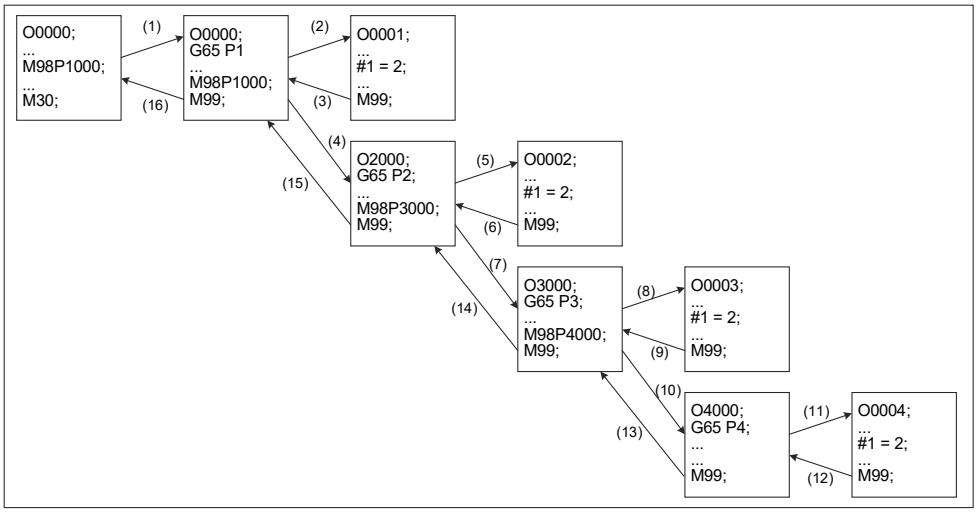
Notes:

(1) G65/G66 can support a maximum of 8 layers of nested calls to the macro program.

(2) If called with the M98 sub-program at the same time, the layers should be separately calculated.

3

Example description:

Example
Example 1: (1) When G65 calls the O1010 macro program, A5.0 and B3.0 are passed to #1 and #2 respectively. (2) When the O1010 macro program value is executed, the program will return to the main program once M99 is encountered. (3) When the G65 L argument is set to 2, and the above action will be repeated twice. (4) When returning to the main program, execution will continue from the N20 block where the G65 called the macro. 
Example 2: G65/G66 can support a maximum of 8 layers of nested calls to the macro program. 

3.11.4.5 Macro modal call (G66/G67)

Macro modal call		group	Command code
Milling		14	G66
Lathe	A-Type		
	B-Type		
	C-Type		

Macro modal call cancel		group	Command code
Milling		14	G67
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G66	P_	L_	I_

G67

P_ Program number

L_ Repeated times

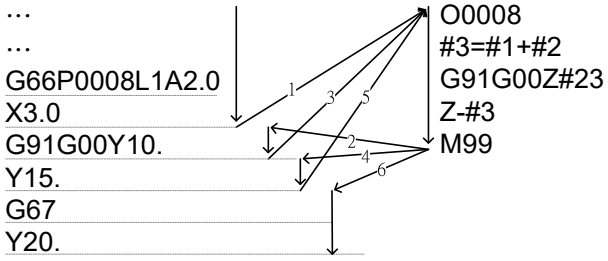
I_ Argument values

Instruction description:

G66 is used to execute the macro program specified by the P argument, and also supports passing arguments to local variables. Unlike G65, the G66 command will enable a command mode in which the system will execute the specified macro after every single block until it is disabled by the instruction G67.

Example description:

Example
The call macro program is started by G66, the O0008 macro will be executed once after the end of each block, and 2.0 is passed to the #1 variable through the A argument; the B argument is not used, so the #2 variable will be NULL. Until G67 cancels this G66 mode, the system starts from the next single block Y20. and will no longer execute the O0008 macro after single blocks execution are completed.



3

3.11.4.6 Interrupt macro call (M96/M97)

Interrupt macro call		Group	Command code
Milling		-	M96
Lathe	A-Type		
	B-Type		
	C-Type		

Interrupt macro call cancel		Group	Command code
Milling		-	M97
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

M96	P_

M97

M96 Interrupt macro call enable.

M97 Interrupt macro call cancel.

P_ Program number.

Instruction description:

The interrupt macro program calling function is enabled by the M96 command.

When the system executes the M96, it will enable the interrupt macro call function and the system will start monitoring and continue to run the main program. If the trigger signal of M2X019 is received during the process, the program currently being executed will be suspended immediately and the specified interrupt macro program will be executed. It will not return to the main program until M99 is used in the macro.

The special relays related to the interrupt macro program MLC are as follows:

Relay address				Function name	Explanation	Property
Path 1	Path 2	Path 3	Path 4			
M21019	M22019	M23019	M24019	Interrupt macro function switch	The special relay to switch the interrupt macro function. 0: Disable. 1: Enable.	R/W
M31031	M32031	M33031	M34031	Interrupt macro function status	The status of the interrupt macro function. 0: Not enabled. 1: Enabled.	R

Notes:

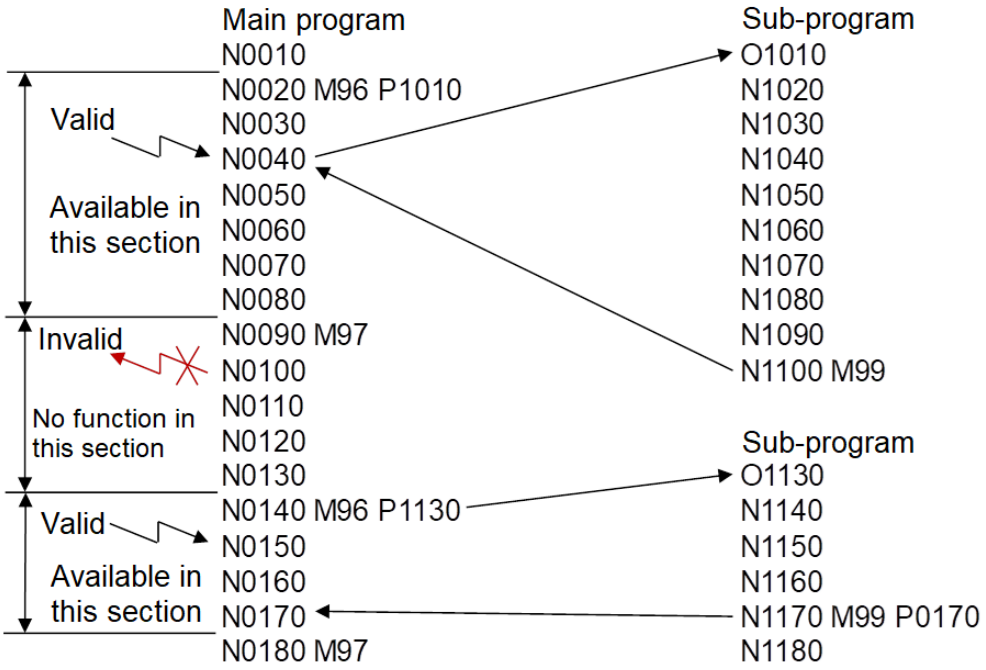
(1) The interrupt macro call function is not supported in the subroutine function.

(2) If the program is returning from the interrupt function and the previous command was G02/G03, the system will move linearly directly to the target position of the G02/G03 and continue running.

(3) The interrupt macro call function is prohibited when tool radius compensation (G41/G42) or scaling (G51) are enabled.

[Example description]

Example



3

3.11.4.7 Sub-program (M98/M99)

Sub-program call		Group	Command code
Milling		-	M98
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

M98	P_	H_
		Q_

P_ Sub-program number, range: 0000-9999.
If no argument is specified, the system will call the current program.

H_ The N label of the sub-program to be started from. Can be omitted.

Q_ The specific line number of the sub-program to be started from. Can be omitted.

Instruction description:

M98 is used to call the subroutine macro number specified by the P argument and can specify that execution starts from a specific line number of the program. It can be used to simplify programing by calling subroutines when there are repeated processes or paths in the program.

Notes:

(1) If the H_ and Q_ arguments are programed at the same time, the H_ will be executed with priority.

(2) If neither the Q_ nor H_ arguments are programed, the system will start from the first block of the sub-program.

(3) If the P_, Q_, and H_ arguments are not programed, the system will issue an alarm [0x261 H_variable is missing].

Back		Group	Instruction code
milling		-	M99
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

M99	P_

P_ Return to specified N label of the original calling program.
If this argument is omitted, the system will start from the next block of the M98 block.

Instruction description:

M99 sub-program call return instruction.

- If the P argument is not programed in the main program, the system will return to the first block for execution.
- If the P argument is programed in the main program, the system will return to the specified N label block for execution.

- If the P argument is not programed in the sub-program, the system will return to the original program of the next single block when the sub-program is called.
- Set the P argument in the sub-program and the system will return to the original program of the specified N label block for execution.

3.11.4.8 T-code call macro

The T code is the system tool replacement code. When the program executes the T code, an MLC signal will be activated to perform the tool replacement process; if the parameter **[N1.010 Bit22 T code call macro O9000]** function is enabled, the system will execute the O9000 macro when any T code is executed in the program. The T code which is in the O9000 will execute as regular T function without calling O9000 macro.

Note:

(1) The T code must be at the beginning of the block when using T code call macros.

(2) The M and S code in the block will be regarded as arguments when using T code call macros.

(3) The M code call macro function is prohibited in the block when using T code call macros.

(4) When executing a macro set of T code calls, both M and T codes are handled by the MLC.

3.11.4.9 G-code call Macro

Aside from the standard G code command function, the G code instruction can also be set as a call macro function. When macro is called by G code, the same G code argument in the macro will be executed as a standard command and no longer has the macro call function. The G code call macro program includes continuous and single call modes. When a G code number of sets both functions at the same time, only the single call is valid.

(1) Continuous mode related parameters:

[N1.120 Beginning G code of G macro call].

[N1.121 Beginning macro number of G macro call].

[N1.122 Amount of G macro call] sets the continuous G code call macro function.

Please refer to [NC5 Maintenance and operation manual] for a detailed description.

(2) Single call mode parameters:

Set the G code value in parameters **[N1.200]** to **[N1.249]**. The corresponding macro between O9100 and O9149 will be executed when the G code argument is executed. For example: the value of N1.200 is 100. When the system reaches the G100 instruction, the system will call the O9100 macro.

Please refer to [NC5 Maintenance and operation manual] for a detailed description.

Note:

(1) The G code must be at the beginning of the block when using G code call macros.

(2) The M, S and T code in the block will be regarded as arguments when using G code call macros.

(3) The M and T code call macro functions are prohibited in the block when using G code call macros.

(4) When the executed G code is in the macro set of G code call, the G code function will be set according to the settings of the parameter **[N1.010 Bit28 Allow G code call in multilayer]**.

0: Default setting allows G code call functions to be performed on a single layer.

1: The G code call function allows multilayer calls; the maximum is 8 layers.

3.11.4.10 M-code call Macro

M code instructions can be divided into two modes. The first is the auxiliary function control, which will activate an MLC signal to perform the auxiliary function switching machine behavior. The other function is the call macro, so that the specified macro will be executed when the M code is executed in the program. The M code call macro program includes continuous and single call modes. When an M code number of sets both functions at the same time, only the single call is valid. If the M code is already in the call macro, its M code will be executed as a standard function and will no longer have the call macro function.

(1) Continuous mode related parameters:
[N1.123 Beginning M code of M macro call].
[N1.124 Beginning macro number of M macro call].
[N1.125 Amount of M macro call] sets the continuous M code call macro function.
Please refer to [NC5 Maintenance and operation manual] for a detailed description.

(2) Single call mode parameters:
Set the M code value in parameters **[N1.250] to [N1.299]**. The corresponding macro between O9150 and O9199 will be executed when the M code argument is executed. For example: the value of N1.250 is 100. When the system reaches the M100 instruction, the system will call the O9150 macro.
Please refer to [NC5 Maintenance and operation manual] for a detailed description.

Note:

(1) The M code must be at the beginning of the block when using M code call macros.
(2) The S and T code in the block will be regarded as arguments when using M code call macros.
(3) The T code call macro function is prohibited in the block when using M code call macros.
(4) When executing a macro set of M code calls, the M, S and T codes are handled by the MLC.

3.12 Compensation function

3

3.12.1 Tool radius compensation (G40/G41/G42)

Tool radius compensation cancel		Group	Command code
Milling		07	G40
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G40

Tool radius left compensation		Group	Command code
Milling		07	G41
Lathe	A-Type		
	B-Type		
	C-Type		

Tool radius right compensation		Group	Command code
Milling		07	G42
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

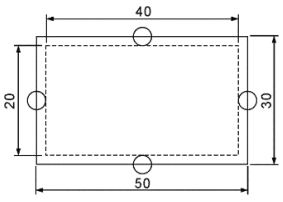
G41	D_
G42	

D_ Tool radius compensation number, expressed as a 2-digit number.

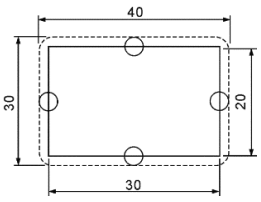
Instruction description:

Since there are many types and sizes of cutting tools used on processing machines, for convenience, most of the programming of general processing paths does not consider the parameters of tools. Instead, they directly describe the outline of the finished product. Furthermore, this approach makes checking the processing path more intuitive.

However, the tool center will cut along the contour of the workpiece when the programmed path does not have tool radius compensation. In fact, such a machining path will cause the finished product size to be reduced by an additional cutting amount of one tool radius due to the radius of the tool. This is shown in the figure below (tool diameter 10 mm):



Milling without outer radius compensation



Milling without inner radius compensation

It can be seen from the above that if the tool mills along the contour of the workpiece, since the tool has a certain diameter, the size of the milling result will be reduced by the radius of the tool.

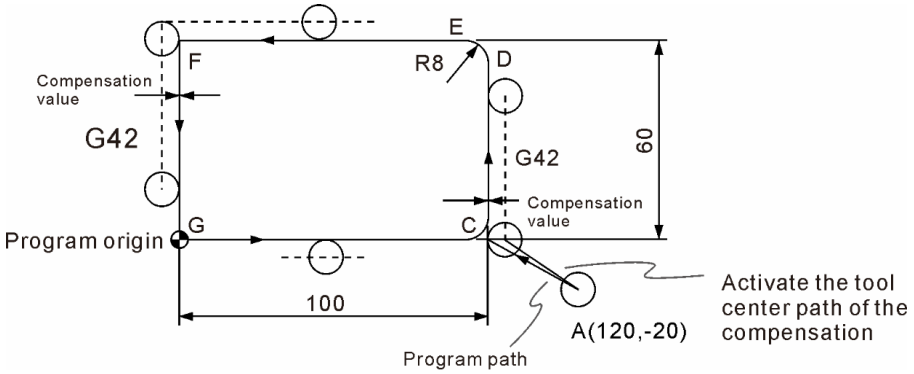
The tool radius compensation function allows the user to select the compensated tool radius size from the tool magazine, and the controller will automatically plan the tool radius compensation path, which can be divided into tool radius left compensation and right compensation. The judgment of the left and right directions is based on the following definition: based on the machining direction, if the tool is on the right side of the workpiece, the tool radius compensation should be compensated to the right with the G42 command; otherwise, if the tool is on the left side of the workpiece, the tool radius compensation should be compensated to the left with the G41 command.

The argument D is the tool radius number in the G41/42 command and represented by a 2-digit number. The controller will read the tool radius data of the OFS group function as the compensation value based on the number specified by D. For example, D11 means using the number 11 tool radius compensation. If the data is 4.0, it means the tool radius is 4.0mm.

When the tool path does not require tool radius compensation, the G40 command can cancel the compensation. Since the compensation command is a continuous command, the tool radius compensation function will remain continuously valid if the function cancellation is not executed. However, when the reference point return command is executed, the tool radius compensation function will temporarily cancel the compensation and resume the compensation function in the next motion block. In addition, the first motion block after canceling tool radius compensation must be a linear motion command.

Note:

(1) When the radius compensation command is enabled, it can be programmed in the same block as G00 or G01, but it cannot be enabled with the G02 or G03 command. If users need to use the tool radius compensation function on an arc path, the tool radius compensation function must be enabled on a linear motion command in advance. When the system executes the tool radius compensation command of G41/G42, it will only update the relevant coordinate system, and the actual movement will take effect on the next motion command. An example of the tool radius right compensation is illustrated below:



Move from point A to point C and enable the tool right compensation instruction as follows:

N00 G90 G00 X120.0 Y-20.0	(Rapid positioning to point A)
N01 G01 G42 X100.0 Y0 D20 F80	(A point to C point)
N02 Y52.0	(C point to D point)
N03 G03 X92.0 Y60.0 R8.0	(D point to E point)
N04 G01 X0	(E point to F point)
N05 Y0	(F point to G point)
N06 X100.0	(G point to C point)

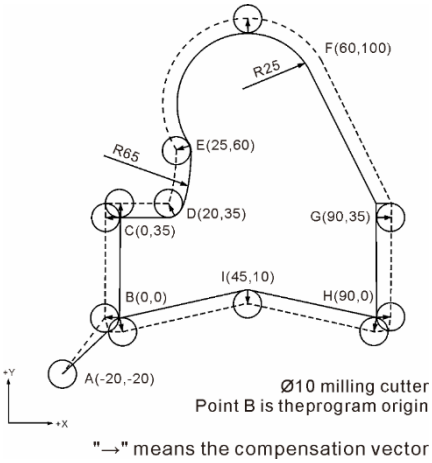
3

(2) Each of the tool radius compensation numbers in the program, such as D11, D12, etc., will correspond to a tool magazine. This value must be filled in the OFS group by the operator in advance.

(3) When the sign of the compensation value changes, the directions of G41 and G42 change accordingly. For example: when the G41 command gives a positive value, the compensation is to the left; if a negative value is given, the compensation is to the right. Similarly, when G42 is given a positive value, its compensation will be to the right; if it is given a negative value, its compensation will be to the left.

(4) When the reference return command of G28 or G29 is executed, the tool radius compensation will be temporarily canceled, and the compensation state will be automatically restored in the next motion block after the return function is completed.

(5) When G40 is executed to cancel the compensation motion command, the compensation will be canceled in the motion path in the opposite direction of the left or right compensation. Therefore, when using G40, it is best to cancel tool radius compensation after the tool has moved away from the workpiece. See the following program for an example:

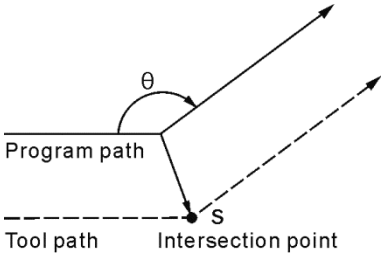
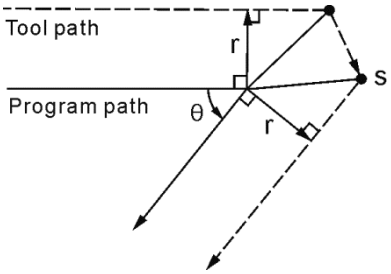
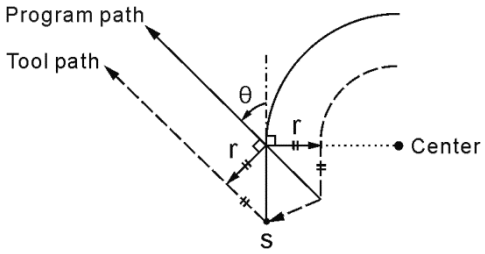
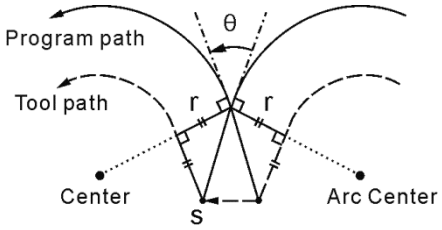



```
N00 G90 G00 X-20.0 Y-20.0
(Rapid positioning to point A)

N01 G01 G41 X0 Y0 D12 F80
(A point to B point, enable left compensation)

N02 Y35.                    (B point to C point)
N03 X20.                    (C point to D point )
N04 G03 X25.0 Y60.0 R65.0   (D point to E point)
N05 G02 X60.0 Y100.0 R25.   (E point to F point)
N06 G01 X90.0 Y35.0         (F point to G point)
N07 G01 Y0                  (G point to H point)
N08 X45. Y10.               (H point to I point)
N09 X0 Y0                   (I point to B point )
N10 X -20. Y -20.           (B point to A point)
N11 G40
(Tool disengaged, cancel compensation)
```

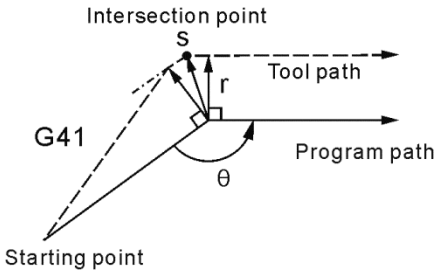
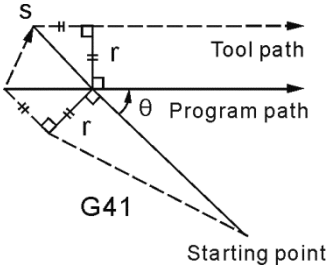
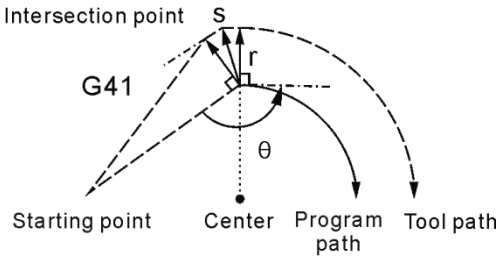
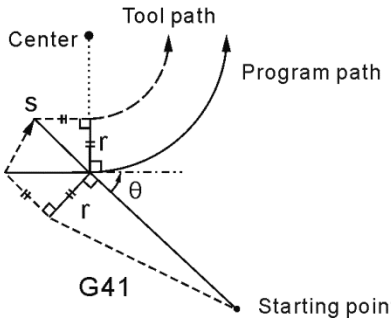
(6) Aside from using G40 to cancel the tool radius compensation, when the program reaches the last motion block and there are no further motion blocks, the last motion block of the entire program will not have the tool radius compensation function.

G41/G42 tool radius compensation path:

The tool radius compensation path is divided into obtuse angles (90°<θ<180°) and acute angles (0<θ<90°) according to the angle formed between the single blocks. When the turning angle is an obtuse angle, the tool path moves in an inner corner; when the turning angle is an acute angle, the tool path moves in an outer corner. As shown below.

Linear line with inside angle	Linear line with outside angle
	
Arc connected to linear line with outside angle	**Arc connected to arc with outside angle**
	

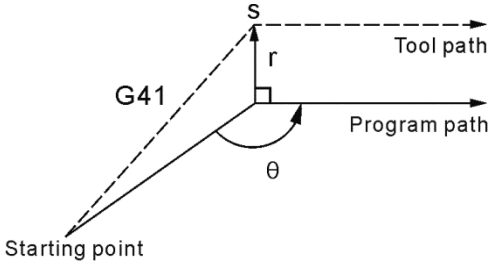
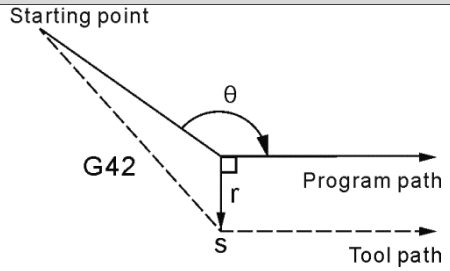
After the tool radius compensation G41/42 is enabled, the actual tool radius compensation movement will be compensated by a motion command until the tool radius compensation is turned off. The first connecting path when it is enabled is as shown in the figure below:

Linear line connects	
	
Linear line connects with arc	
	

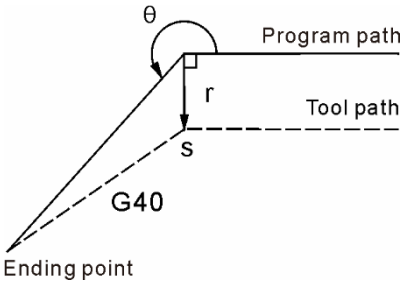
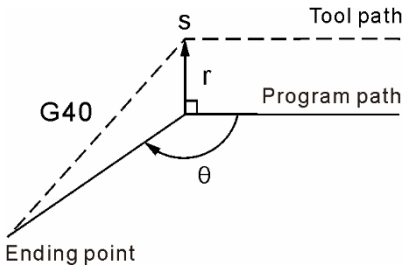
3

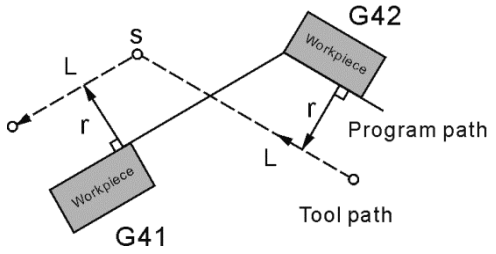
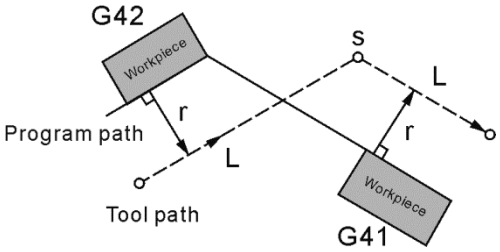
Compensation path exchange:

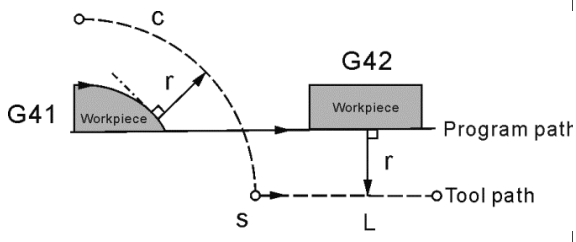
When entering the compensation path from a motion path without compensation, the tool center motion path is as shown in the figure below:

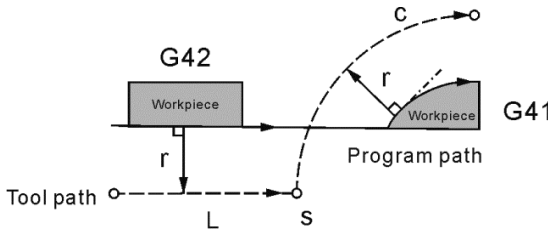
G40 to G41	G40 to G42
	

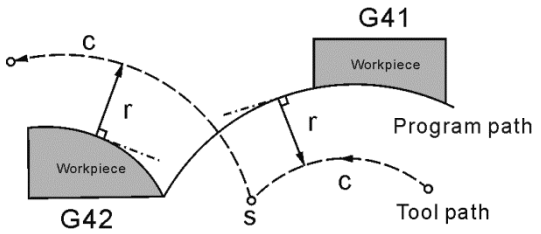
When G40 is used to cancel the compensation or when the compensation direction is switched, the tool center motion path is as shown below:

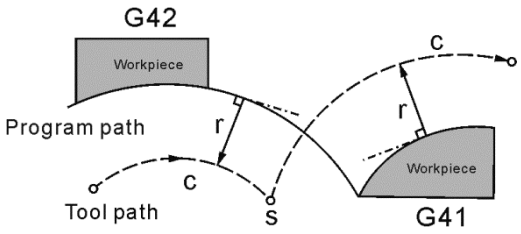
G41 to G40	G42 to G40
	

G41 to G42	G42 to G41
	









3.12.2 Tool length compensation (G43/G44/G49)

Tool length positive compensation		Group	Command code
Milling		08	G43
Lathe	A-Type		
	B-Type		
	C-Type		

Tool length negative compensation		Group	Command code
Milling		08	G44
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G43 / G44	CP	H_	Z_

Z_ Z axis reference position. The length compensation is based on this Z-axis position.

H_ Tool length compensation number, expressed by a 2-digit number.

Tool length compensation cancel		Group	Command code
Milling		08	G49
Lathe	A-Type		
	B-Type		
	C-Type		

Instruction format:

G49

Instruction description:

In the process of machine processing, many tools with different lengths may need to be used. Therefore, this command function enables the use of the program to specify tool length data numbers. The controller will perform height compensation on the tool length data to different tool lengths, which will maintain the tool tip point at the same height as the command position. The tool length compensation function is a continuous command.

After executing the tool length compensation function, the system will activate the length value until a new compensation number is issued. Therefore, before selecting a new tool, it is better to cancel the previous tool length compensation or execute the tool length compensation command corresponding to the tool once after tool changing.

The tool length compensation data number is represented by a 2-digit number, and the system will use this tool length number in the data table for compensation. For example, H01 means that the system will use the tool length number 01 when executing the G43 or G44 command. When the tool magazine in the OFS is registered number 01 as -412.8, the tool length compensation value of H will be -412.8 mm. The G43 tool length compensation will be upward

3

when the compensation value is a positive value; if it is a negative value, the tool length compensates downward. Similarly, if G44 is programed as a positive value, the tool will compensate downward, and if it is a negative value, the tool will compensate upward.
After executing the G49 command, the tool length compensation function is canceled.

Note:

(1) G43, G44 are continuously valid functions, and can only be canceled by the G49 or H00 instructions.
(G49 is tool length compensation cancel, and H00 means the compensation value is zero)

(2) When executing commands such as G53, G28 or G30, the system will automatically cancel the tool length compensation value. If the H_ of G43/G44 is not specified again, the system will continue to process without tool length compensation.

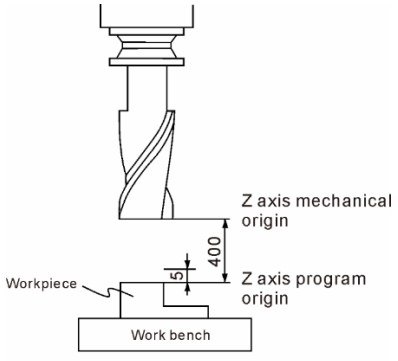
(3) Parameter **[N1.010]** is used to set whether the compensation positioning action will be performed in the G43, G44 or G49 tool length compensation block. When the setting value is 0, it means the system will move the tool to the compensated position at the tool length compensation block that does not contain a Z command; when the setting value = 1, it means the system will only process internally at the tool length compensation block that does not contain a Z command without any movement.

(4) When executing the G28/G30 reference point return command, the system will move to the middle point with tool length compensation, but when it returns to the reference position, the tool length compensation function will be automatically canceled, and the following blocks will not automatically restore tool length compensation.

(5) When one of program end commands M30 or M02 are executed, the system will automatically cancel the tool length compensation and return to the G49 system state. When the system receives the RESET command, the system will automatically cancel the tool length compensation and return to the G49 system state.

(6) When the system receives the RESET command, the system will automatically cancel the tool length compensation and return to the G49 system state.

Example description:

Example	Program / description
 Tool length compensation	The following two command examples have the same results, as shown in the figure on the left: (1) Tool length H01 as -400.000 G43 Z5.0 H01 (2) Tool length H01 as 400.000 G44 Z5.0 H01

3.12.3 Scaling (G50/G51)

Scaling cancel		Group	Command code
Milling		11	G50
Lathe	A-Type		-
	B-Type		G50
	C-Type		G50

Scaling		Group	Command code
Milling		11	G51
Lathe	A-Type		-
	B-Type		G51
	C-Type		G51

Instruction format:

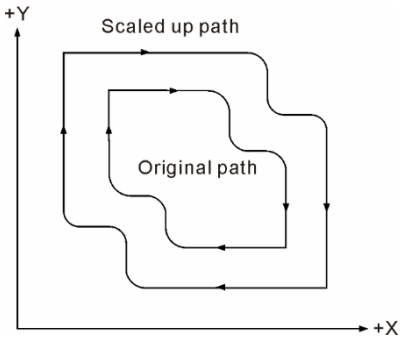
G51	X_ Y_ Z_	I_ J_ K_	P_

X_ Y_ Z_ Scaling coordinate center.

I_ J_ K_ Corresponding X, Y, Z axes scaling ratio.

P_ Total ratio.

Instruction description:



This command can be used in machining programs for products with different size ratios. Using X_ Y_ Z_ as the coordinate center to execute the overall proportion setting P, the proportion of the machining path can be enlarged or reduced. Alternatively, the corresponding ratio of X, Y and Z axis can be individually specified in the I_ J_ K_ format. The minimum command value of the P_ format and the I_ J_ K_ format is 1, and the magnification range is 0.001–999.999. For example: P100 is equivalent to shrinking to 0.1 times.

As shown in the figure above, under the original program proportions, the new path will change according to the set ratio after the scaling ratio P value is set by the scaling command G51. When the scaling function is executed, the compensation of tool radius, length and position will not be affected. The G50 command will restore the original unit ratio and perform the cutting function of the normal path. In addition, when M02, M30 is executed or the NC controller has a reset command, the proportional scaling mode will be canceled.

3

3.12.4 Mirror (G24/G25/G50.1/G51.1)

Mirror enable		Group	Command code
Milling		18	G24
Lathe	A-Type		G51.1
	B-Type		G51.1
	C-Type		G51.1

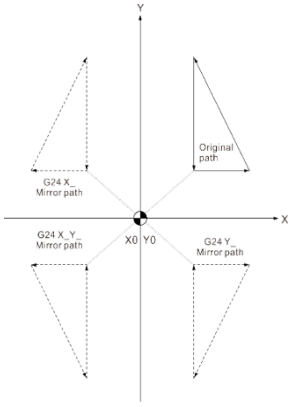
Mirror cancel		Group	Command code
Milling		18	G25
Lathe	A-Type		G50.1
	B-Type		G50.1
	C-Type		G50.1

Instruction format:

G25	CP
G24	

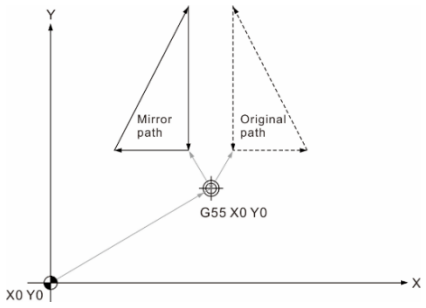
CP Mirror axis direction or center position.

Instruction description:



When executing the scaling command, users can specify any CP coordinate point as the mirror center. The system will generate the new path according to the specified mirror axis. Therefore, when the left and right or upper and lower paths are symmetrical to each other, users can use the mirror function to convert the motion path on the other side, which saves the time that would have otherwise been used for re-programing. When using G25 to cancel mirroring, the mirror axis must also be specified. For example: G25 Y_. Only the Y-axis mirroring function will be canceled, the other axes will still maintain the mirroring function.

Example description:

Example	Program / description	
	N00G00G90G40G49 N01G55X0Y0 N02G01X5.Y10.F1000 N03X35.Y10. N04X5.Y70. N05X5.Y10. N06G00X0Y0 N07G24X0 N08G01X5.Y10.F1000 N09X35.Y10. N10X5.Y70. N11X5.Y10. N12G00X0Y0 N13G25X0 N99M30	The dotted line in the left figure marks the N02–N05 blocks. The N07 line uses the X0 coordinates to turn on the mirror function and execute the same G code N08–N11 path as N02–N05. N13 G25 X0 cancels the mirror function of the X axis, and the motion path of the program returns to the original motion path planning method.

3.12.5 Rotation (G68/G69)

Coordinate rotation		Group	Command code
Milling		15	G68
Lathe	A-Type	-	
	B-Type		
	C-Type		

Instruction format:

G68	X_ Y_	R_

X_Y_ Rotation coordinate center.

R_ Rotation angle. Positive value for counterclockwise rotation and negative for clockwise rotation.

The smallest unit for rotations is 0.001 degrees, and the range is from 0 degrees to 360 degrees.

Coordinate rotation cancel		Group	Command code
Milling		15	G69
Lathe	A-Type	-	
	B-Type		
	C-Type		

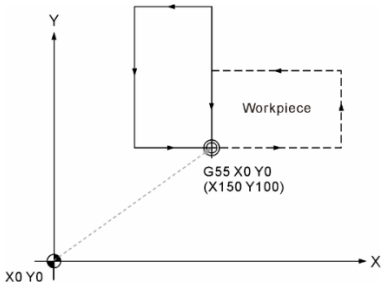
Instruction format:

G69

Instruction description:

The G68 coordinate rotation command can use the specified coordinate position as the rotation center to apply a specified rotation angle to the original program. Therefore, when the workpiece is placed at an angle different from the direction of the original machining program, the G68 command can be used to rotate the coordinate system. This simplifies the complex calculation of program path points and saves the time that would have otherwise been used for re-programing.

Example description:

Example	Program / Description
	Original path: O1000 G00 G90 G55 X0 Y0 G00 X100.0 Y50.0 X0.0 Y0.0 M30 Rotates in 90 degrees: O1000 G00 G90 G55 X0 Y0 G68 X0 Y0 R90.0 G00 X100.0 Y50.0 X0.0 Y0.0 M30 **Description:** The original program is the dotted line graphic on the left. Originally, if the placement direction of the workpiece needs to be changed, the program path would have to be re-programed. The G68 command can be used to rotate at a specific angle, which means the program does not need to be re-programed. The G69 command cancels the coordinate rotation function. When the rotation function is canceled, the coordinate system of the program will return to the original coordinate system.

3.13 Canned cycle instruction – Milling

This section introduces the canned cycled instructions for the milling machine, including drilling, tapping, and boring instructions.

3

3.13.1 Drilling cycle (G73/G81/G82/G83)

Peck drilling cycle	Group	Command code
Milling	09	G73

Drilling cycle	Group	Command code
Milling	09	G81

Drilling with dwell	Group	Command code
Milling	09	G82

Deep hole peck drilling cycle	Group	Command code
Milling	09	G83

Instruction format:

G73	X_	Y_	Z_	R_	Q_	F_	K_

G81	X_	Y_	Z_	R_	F_	K_

G82	X_	Y_	Z_	R_	P_	F_	K_

G83	X_	Y_	Z_	R_	Q_	F_	K_

X_Y_ Target position.

Z_ Drilling depth position.

R_ Initial safe height.

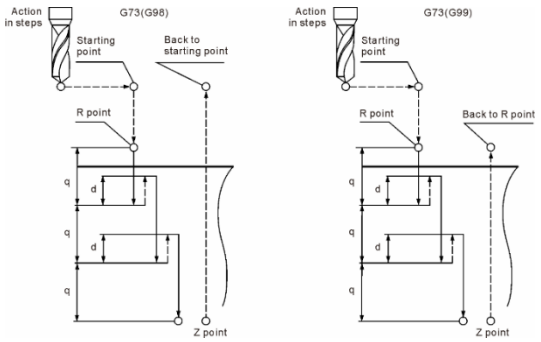
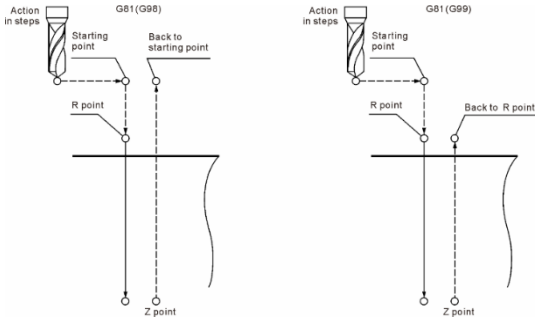
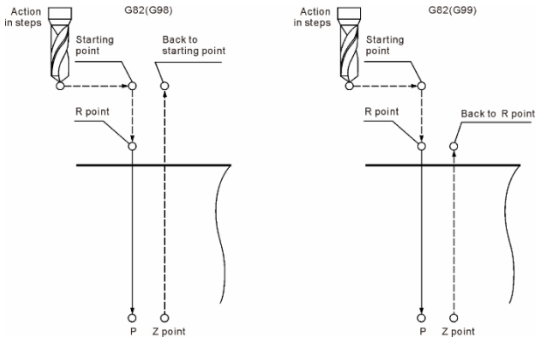
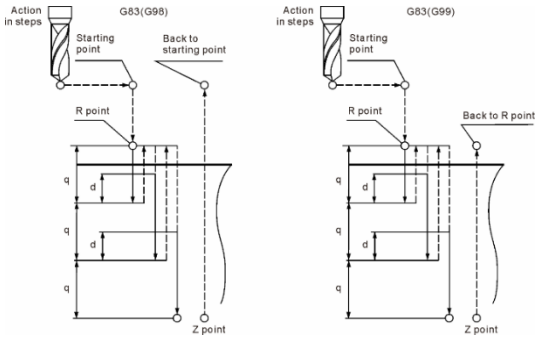
P_ Pause time. Unit: ms

Q_ Single peck drill depth.

F_ Cutting feed speed.

K_ Number of circulating holes.

Instruction description:

Instruction	Example	Description
Peck drilling cycle [G73]		The peck drilling cycle is an intermittent process. The steps are as follows: (1) The Z axis rapidly descends to the safe height R_. (2) The Z axis drills down with the F_ speed by single peck drilling to the depth Q. (3) The Z axis rapidly rises back up to the drilling escape amount. (4) Based on the completed hole bottom, it drills downward to a depth of Q in a single drilling. Cutting is performed at speed F_. (5) Repeat steps 3 and 4 until the Z axis reaches the depth of the hole. (6) The Z axis returns to the specified height.
Drilling cycle [G81]		Drilling cycle instruction. Steps as below: (1) The Z axis rapidly descends to the safe height R_. (2) The Z axis drills down with the F_ speed by drilling to the depth Q. (3) The Z axis returns to the specified height.
Drilling with dwell [G82]		Drilling cycle with counter bore, pause a moment at the bottom. Steps as below: (1) The Z axis rapidly descends to the safe height R_. (2) The Z axis drills down with the F_ speed by drilling to the depth Q. (3) Pause time P_. (4) The Z axis returns to the specified height.
Deep hole peck drilling cycle [G83]		The deep hole peck drilling cycle is an intermittent process with auto return. Steps as below: (1) The Z axis rapidly descends to the safe height R_. (2) The Z axis drills down with the F_ speed by single peck drilling to the depth Q. (3) Z axis returns to the specified height. (4) Based on the completed hole bottom, it drills downward to a depth of Q in a single drilling. Cutting is performed at speed F_. (5) Repeat steps 3 and 4 until the Z axis reaches the depth of the hole. (6) The Z axis returns to the specified height.

3

Note:

(1) The distance d of the G73 and G84 single Z axis withdraw amount is specified by the parameter **[N1.151 Peck-drilling withdraw distance]**.

(2) In the drilling cycle command state, the tool radius compensation function is ignored.

(3) The G80 command must be executed to cancel the motion state of cycle cutting.

(4) The offset distance Q in the drilling cycle command only supports positive values. If the value is negative, the absolute value will be applied.

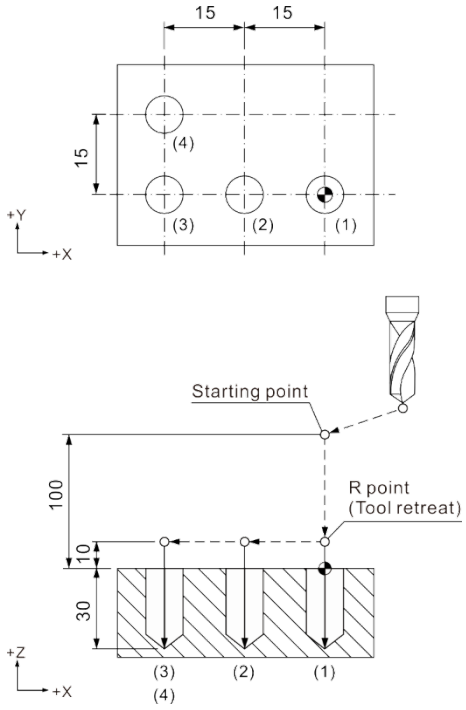
(5) The X_, Y_, Z_, R_, P_, Q_, and F_ arguments in the boring cycle instruction can all inherit the previous status.

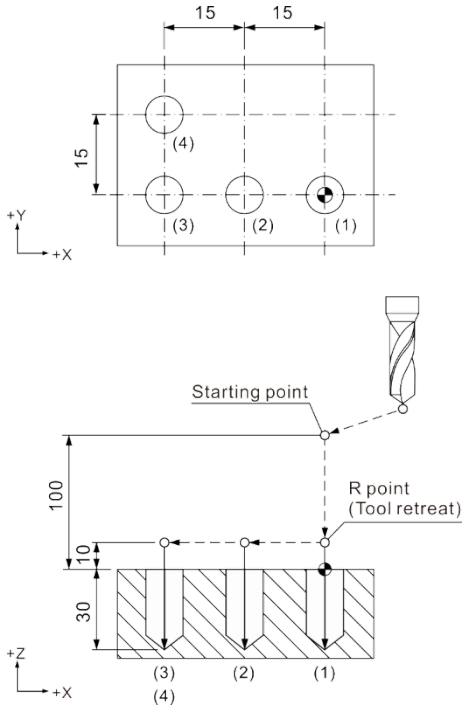
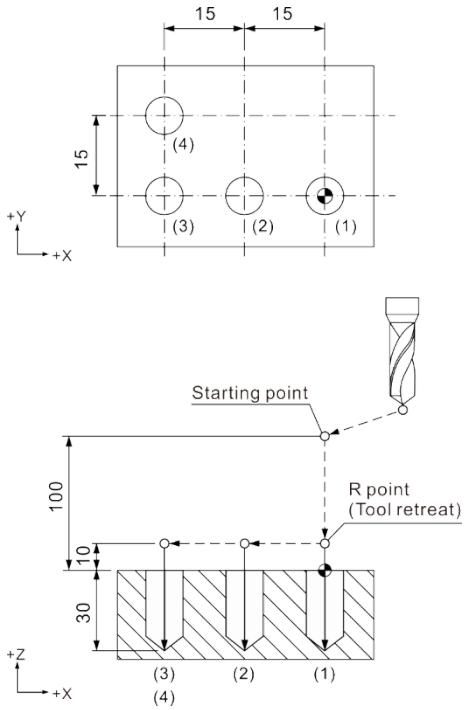
(6) The K_ argument in the loop instruction is valid for only a single block.

(7) If the number of cycle holes K contains a decimal point, the value will be rounded down unconditionally. If K is less than 1, the system will be positioning according to the XY axis command and changed to cycle state. However, the drilling cycle will not be executed.

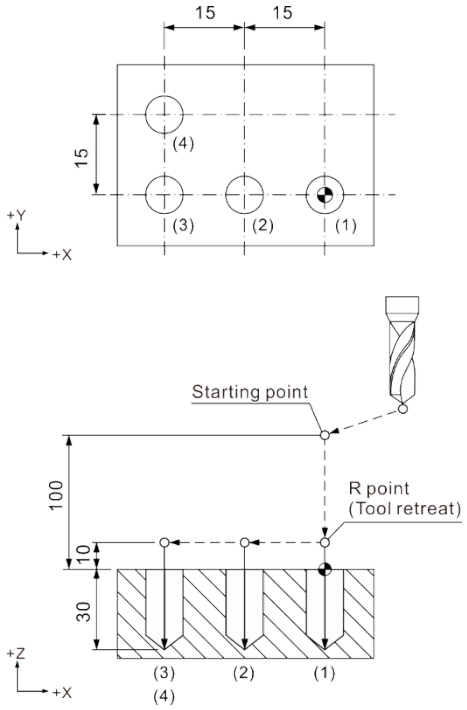
(8) When the system is in the absolute command state, the K value will perform a specified number of cyclic actions at the X_Y_ original position. In the incremental command state, the K value will perform a specified number of incremental equal-distance circular actions based on the specified X_Y_ distance.

Example description:

Example	Program / description
Example 1: 	M03S1000 G17 G90 G00 G54 X0. Y0. G00 Z100. G99 G73 X0. Y0. Z-30. R10. Q4. K1 F100. (1) X-15. (2) X-30. (3) X-30. Y15. (4) G80 G91 G28 X0. Y0. Z0. M05 M30

Example	Program / description
Example 2: 	M03 S1000 G17 G90 G00 G54 X0. Y0. G00 Z100. G99 G81 X0. Y0. Z-30. R10. K1 F100. (1) X-15 (2) X-30 (3) X-30. Y15. (4) G80 G91 G28 X0. Y0. Z0. M05
Example3: 	M03 S1000 G17 G90G00G54X0.Y0. G00 Z100. G99 G82 X0. Y0. Z-30. R10. P100 F100. (1) X-15. (2) X-30. (3) X-30. Y15. (4) G80 G91 G28 X0. Y0. Z0. M05

3

Example	Program / description
Example 4:  	M03 S1000 G17 G90 G00 G54 X0. Y0. G00 Z100. G99 G83 X0. Y0. Z-30. R10. Q4 K1 F100. (1) X-15. (2) X-30. (3) X-30. Y15. (4) G80 G91 G28 X0. Y0. Z0. M05

3.13.2 Tapping cycle (G74/G84)

Left hand tapping cycle	Group	Command code
Milling	09	G74

Right hand tapping cycle	Group	Command code
Milling	09	G84

Instruction format:

G74 G84	X_	Y_	Z_	R_	Q_	S_	F_	K_

X_ Y_ Target position.

Z_ Tap depth position.

R_ Initial safe height.

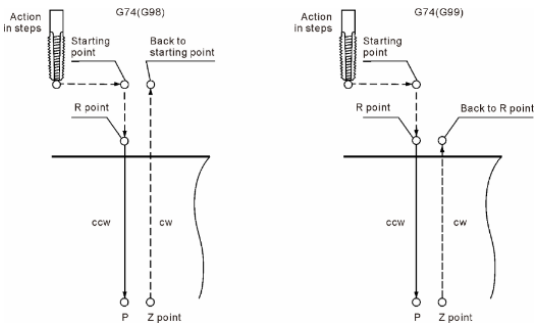
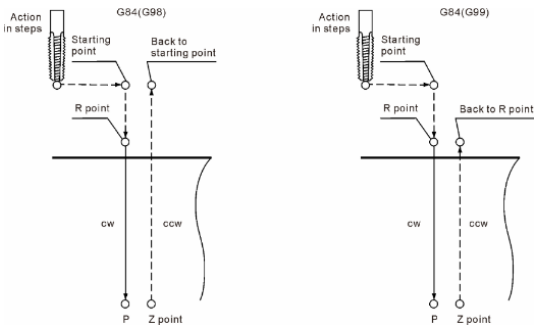
Q_ Single tap depth.

S_ Tapping speed.

F_ Threading speed.

K_ Number of circulating holes.

Instruction description:

Instruction	Example	Explanation
Left hand tapping cycle [G74]		The steps are as follows: (1) The Z axis rapidly descends to the safe height R_. (2) The spindle reverses its rotation, and the Z axis performs tapping with speed F_. (3) After reaching the depth specified by Z, the spindle switches to positive rotation, and the Z axis retreats in the positive direction to the safe height R. (4) The spindle is then restored to the ready state. (5) The Z axis returns to the specified height.
Right hand tapping cycle [G84]		The steps are as follows: (1) The Z axis rapidly descends to the safe height R_. (2) The spindle positive its rotation, and the Z axis performs tapping with speed F_. (3) After reaching the depth specified by Z, the spindle switches to reverses rotation, and the Z axis retreats in the positive direction to the safe height R. (4) The spindle is then restored to the ready state. (5) The Z axis returns to the specified height.

3

Note:

(1) The tapping feed mode is set by the parameter **[N1.150 Bit2-3 Tapping Mode]**.

(2) The X_, Y_, Z_, R_, P_, Q_, and S arguments in the tapping cycle instruction can all inherit the previous status.

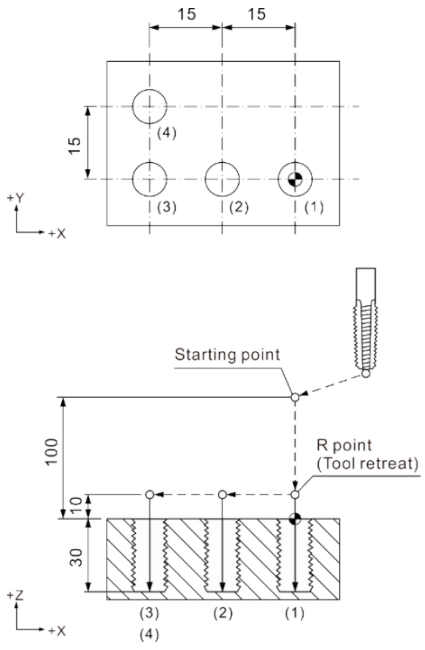
(3) In the tapping cycle command, F_ is a necessary parameter in the drilling cycle block. Otherwise, the alarm [0X1001 Feed rate not specified] will be triggered.

(4) The K_ argument in the loop instruction is valid for only a single block.

(5) Calculation of tapping pitch, lead and feed speed.
G94: Feed rate F (mm/min) = lead P (mm/rev) x spindle speed S (rev/min).
G95: Feed rate F (mm/min) = Lead P (mm/rev)

(6) If an unexpected interruption occurs in the tapping process, including reset, emergency stop, or power interruption, the system will record the current tapping execution data and return the alarms [0x0270 Axis movement prohibition during tapping interruption] and [0x0C09 Spindle movement prohibition during tapping interruption]. After the status is eliminated, the G84.48 command can be executed through the MDI mode to command the spindle and Z axis to exit in reverse and return to the tapping R point; also, M3X708 can be triggered to clear the lock status. However, if the system is in the incremental command mode, the G84.49 cannot ensure that it will correctly return to point R because the servo drive will have lost its position when the system powered off.

Example description:

Example	Program / description
	G17 G90 G00 G54 X0. Y0. G00 Z100. M29 S1000 G99 G74 X0. Y0. Z-30. R10. P1000 K1 F1000. (1) X-15. (2) X-30. (3) X-30. Y15. (4) M28 G91 G80 G28 X0. Y0. Z0.

3.13.3 Boring/reaming cycle (G76/G85/G86/G87/G88/G89)

Fine boring cycle	Group	Command code
Milling	09	G76

Reaming cycle	Group	Command code
Milling	09	G85

Rough boring cycle	Group	Command code
Milling	09	G86

Back boring cycle	Group	Command code
Milling	09	G87

Boring cycle with dwell	Group	Command code
Milling	09	G88

Back boring cycle with dwell	Group	Command code
Milling	09	G89

Instruction format:

G76	X_	Y_	Z_	R_	Q_	F_	K_

G85	X_	Y_	Z_	R_	F_	K_

G86	X_	Y_	Z_	R_	F_	K_

G87	X_	Y_	Z_	R_	Q_	P_	F_	K_

G88	X_	Y_	Z_	P_	F_	K_

G89	X_	Y_	Z_	R_	P_	F_	K_

X_ Y_ Target position. If not specified, the value will be inherited from the previous state.

Z_ Boring depth position. If not specified, the value will be inherited from the previous state.

R_ Initial safe height. If not specified, the value will be inherited from the previous state.

P_ Pause time. Unit: ms. If not specified, the value will be inherited from the previous state.

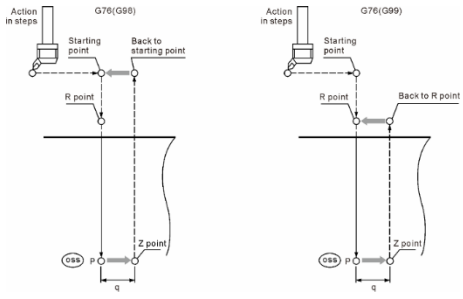
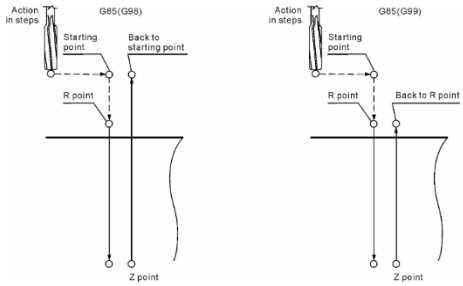
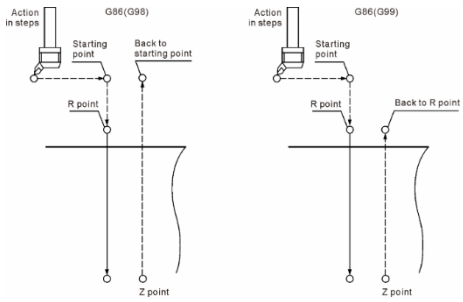
Q_ Offset distance. If not specified, the value will be inherited from the previous state.

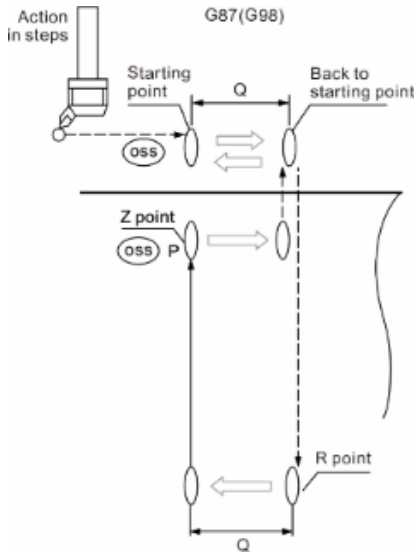
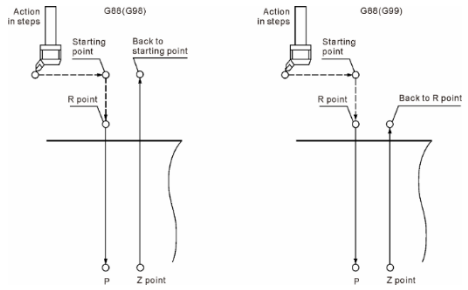
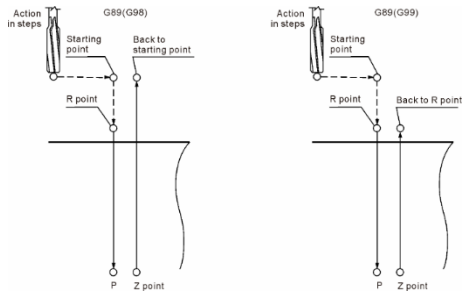
F_ Cutting feed. If not specified, the value will be inherited from the previous state.

K_ Circulation repeat times. Only valid in one block.

3

Instruction description:

Instruction	Example	Program/ Description
Fine boring cycle [G76]		The steps are as follows: (1) Rapid positioning to X_Y_ position. (2) Z axis rapid descent to safe height R_. Boring down with F_ speed to the depth Z. (3) Spindle keeps rotating for a time P_. (4) Spindle stops rotating and performs positioning. (5) The tool disengages from the surface to the specified distance Q_. (6) The Z axis returns to the specified height. (7) X_Y returns to the specified position. (8) The spindle reverts to the previous rotation status.
Reaming cycle [G85]		Steps as below: (1) Rapid positioning to X_Y_ position. (2) Z axis rapid descent to safe height R_. Boring down with F_ speed to the depth Z. (3) Spindle keeps rotating and Z axis return to the save height R with cutting feed F_. (4) The Z axis returns to the specified height.
Rough boring cycle [G86]		The tool in this cycle command is in direct contact with the processing surface. Therefore, when the tool is lifted, there will be slight scratches on the hole wall because the tool is not rotating. This cycle command is usually used for rough boring processing. Steps as below: (1) Rapid positioning to X_Y_ position. (2) Z axis rapid descent to the safe height R_. Boring down with F_ speed to the depth Z. (3) Spindle stop rotating and Z axis return to the save height R with rapid seed. (4) The Z axis returns to the specified height.

Instruction	Example	Program/ Description
Back boring cycle [G87]		The G87 instruction doesn't support G99 mode. Steps as below: (1) Fast positioning to X_Y_ position. (2) After the spindle is positioned to the fix direction of the tool tip, the tool center retreats to the specified distance Q_. (3) Z axis rapid descent to safe height R_, and the center of tool positioning to specified X_Y_ position. (4) The spindle positive rotating, and Z axis boring to specified height Z_ with cutting speed F_. (5) The spindle stops rotating, and the tool center returns to offset Q with spindle positioning at the same time. (6) The tool returns to the Z axis initial position. (7) Tool center positioning to specified X_Y_ position.
Boring cycle with dwell [G88]		Steps as below: (1) Fast positioning to X_Y_ position. (2) Z axis rapid descent to safe height R_. (3) Z axis boring down with cutting feed F_ to the depth position Z_. (4) Spindle spinning for pause time P_. (5) Spindle stops rotating and enter the M00 pause status. (6) Switch to MPG mode and retrace Z axis manually. (7) Switch auto mode and then press cycle start button, Z axis returns to specified height.
Back boring cycle with dwell [G89]		Steps as below: (1) Fast positioning to X_Y_ position. (2) Z axis rapid descent to safe height R_. (3) Z axis boring down with cutting feed F_ to the depth position Z_. (4) Spindle spinning for pause time P_. (5) Z axis returns to safe height R with cutting feed F_. (6) Z axis returns to the specified height.

Note:

(1) When the fine boring cycle command is executed, the M29 will be issued if the Z axis depth is reached and then the MLC must take over the positioning execution.

(2) The offset distance Q in the boring cycle command must be a positive value. If the value is negative, the absolute value will be converted and applied.

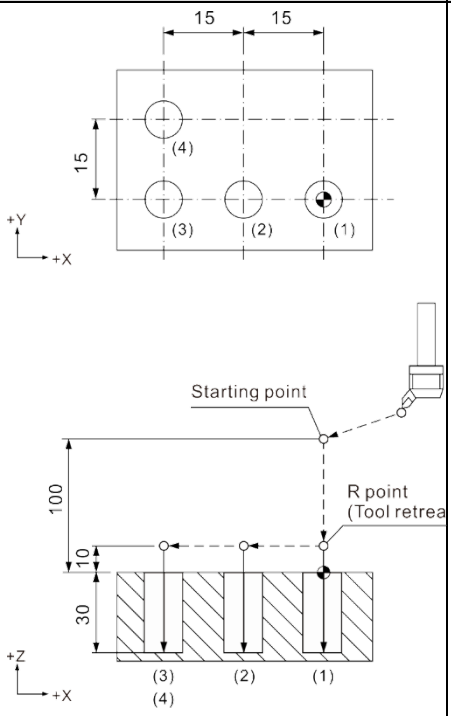
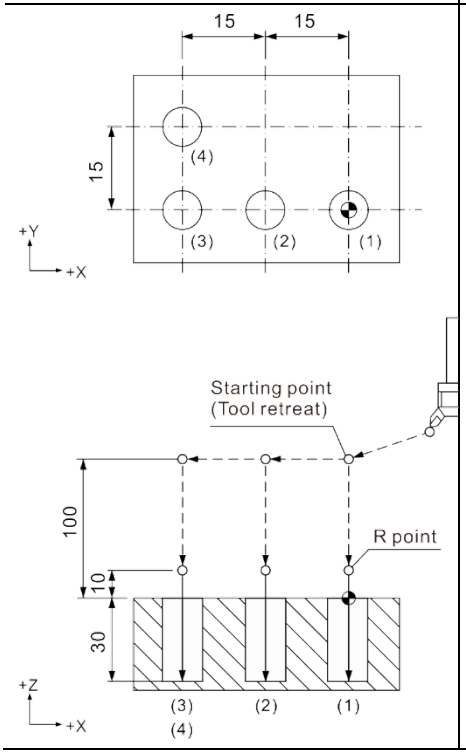
(3) The X_, Y_, Z_, R_, P_, Q_, and F_ arguments in the boring cycle instruction can all inherit the previous status.

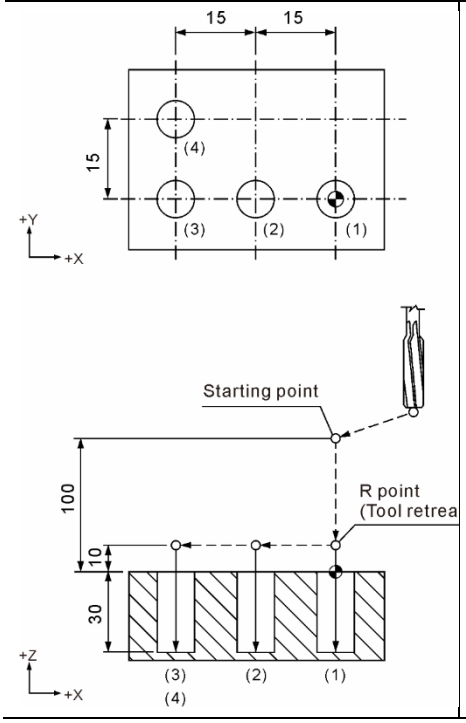
(4) The K_ argument in the loop instruction is valid for only a single block.

(5) The tool retraction offset direction is set by the parameter **[N1.150 Bit0-1 Setting the direction of Q distance in G76 cycle command]**.

(6) Q in the canned cycle instruction is the status value, which includes but is not limited to the single cutting of G73/83 and the offset of G87. An appropriate Q value should be specified to avoid incorrect processing interference when using the canned cycle instruction.

Example description:

Example	Program/ Description
	M03 S1000 G17 G90 G00 G54 X0. Y0. G00 Z100. G99 G76 X0. Y0. Z-30. R10. P1000 Q5. F100. (1) X-15. (2) X-30. (3) X-30. Y15. (4) G80 G91 G28 X0. Y0. Z0. M05
	M03 S1000 G17 G90 G00 G54 X0. Y0. G00 Z100. G98 G76 X0. Y0. Z-30. R10. P1000 Q5. F100. (1) X-15. (2) X-30. (3) X-30. Y15. (4) G80 G91 G28 X0. Y0. Z0. M05

Example	Program/ Description
	M03 S1000 G17 G90 G00 G54 X0. Y0. G00 Z100. G99 G85 X0. Y0. Z-30. R10. F100. (1) X-15. (2) X-30. (3) X-30. Y15. (4) G80 G91 G28 X0. Y0. Z0. M05

3.13.4 Canned cycle cancel (G80)

Canned cycle cancel	Group	Command code
Milling	09	G80

Instruction format:

G80

Instruction description:

The cycle command is a G code group 09 command, and the following blocks of the command will still be used. To cancel the cutting cycle command state, users need to program the G80 cycle cancel command to return to the normal motion mode. The G80 command can cancel the cycle command functions commanded by G73, G74, G76 and G81 - G89.

Example description:

Program / Description	
G17 G90 G00 G54 X0. Y0.	
Z100.	
G99 G73 X0. Y0. Z-20. R10. Q4. K1 F100.	(Execute peck drilling circulation instruction at X0.Y0.)
X10.Y0.	(Execute peck drilling circulation instruction at X0.Y0.)
X20.Y0.	(Execute peck drilling circulation instruction at X0.Y0.)
G80	(Cancel G73 circulation)
G17 G90 G00 G54 X0. Y0.	

3.13.5 Return point of canned cycle (G98/G99)

Return to initial point in canned cycle	Group	Command code
Milling	10	G98

Return to R point in canned cycle	Group	Command code
Milling	10	G99

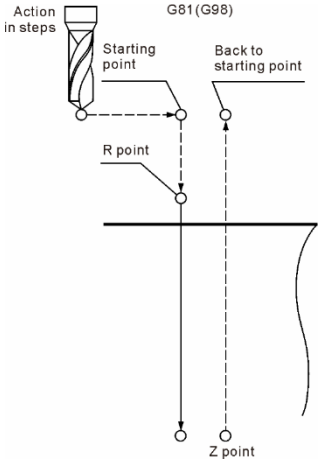
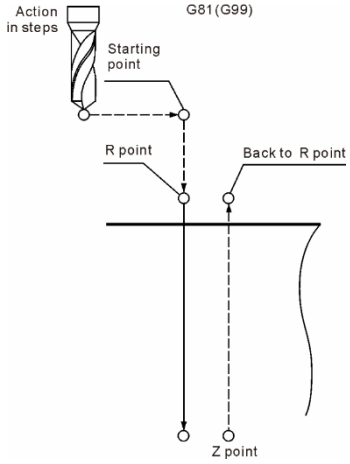
Instruction format:

G98
G99

Instruction description:

G98/G99 is a motion command used to return the tool to the specified height after the fixed cycle command is completed. When the G98 command is specified, the tool will return to the starting point of the fixed cycle command; when the G99 command is specified, the tool returns to the reference point R after the cycle command is completed.

Program example:

G98 return to initial point in canned cycle	G99 return to R point in canned cycle
	

3.14 Cycle instruction - Lathe

3

3.14.1 Drilling cycle (G83/G87)

End face drilling cycle		Group	Command code
Lathe	A-Type	09	G83
	B-Type		G83
	C-Type		G83

Side drilling cycle		Group	Command code
Lathe	A-Type	09	G87
	B-Type		G87
	C-Type		G87

Instruction format:

G83	X_	C_	Z_	R_	Q_	P_	F_	K_	L_
G87	U_	H_	W_						

X_/U_ The G83 instruction is for the X drilling coordinate.
The G87 instruction is for the hole bottom coordinate.

C_/H_ C axis drilling coordinate.

Z_/W_ The G83 instruction is for the hole bottom coordinate.
The G87 instruction is for the Z drilling coordinate.

R_ Reference point position.

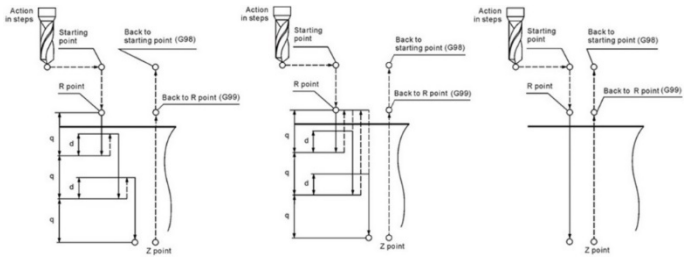
Q_ Pitch.

P_ Bottom pause time.

F_ Feed speed.

K_ Repeated times.

L_ Drilling mode selection.



Instruction description:

The end face and side drilling command can select the drilling mode through the argument L. If L is not specified, the system will refer to parameter setting **[N1.150 Bit2-3 Tapping mode]**:

0: Normal.

1: Deep hole peck. Argument Q is the pitch on each cycle and R is the return position.

2: Peck. Argument Q is the pitch on each cycle, using **[N1.151 Peck-drilling withdraw distance]**.

The action is processed based on the command selection to position the hole and drilling axis. The axes and corresponding commands are as follows:

	G83	G87
Drilling axis	Z	X
Positioning axis	X, C	Z, C

Note:

(1) When the lathe G code is set to A type, the arguments U_, H_ and W_ are for incrementing the values of the X, C and Z axis-direction coordinates.

(2) When the lathe G code is set to B or C type, G90 and G91 are used for switching incremental modes.

3.14.2 Tapping cycle (G84/G88)

End face tapping cycle		Group	Command code
Lathe	A-Type	09	G84
	B-Type		G84
	C-Type		G84

Side tapping cycle		Group	Command code
Lathe	A-Type	09	G88
	B-Type		G88
	C-Type		G88

Instruction format:

G84	X_	C_	Z_	R_	Q_	P_	F_	K_	L_
G88	U_	H_	W_						

X_/U_ The G84 instruction is for the X tapping coordinate.
The G88 instruction is for the hole bottom coordinate.

C_/H_ C axis tapping coordinate.

Z_/W_ The G84 instruction is for the hole bottom coordinate.
The G88 instruction is for the Z tapping coordinate.

R_ Reference point position.

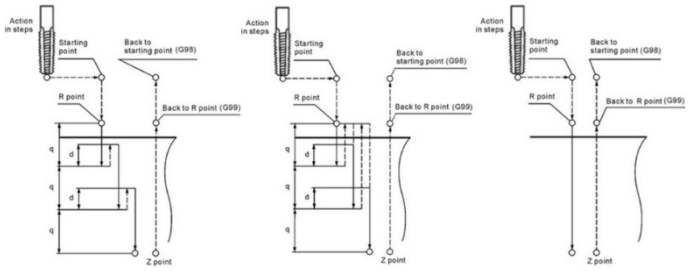
Q_ Pitch.

P_ Hole bottom pause time. Unit: ms.

F_ Feed speed.

K_ Repeated times.

L_ Tapping modes selection.



Instruction description:

In the tapping cycle command, the spindle will first position to the Z-phase mark and then rapidly move to the reference position R. After the spindle rotates and performs the tapping action to the set hole bottom position, it reverses and retracts to point R, and then rapidly moves to the initial point.

The end face and side drilling command can select the drilling mode through the argument L. If L is not specified, the system will refer to parameter setting **[N1.150 Bit2-3 Tapping mode]**:

0: Normal.

1: Deep hole peck. Argument Q is the pitch on each cycle and R is the return position.

2: Peck. Argument Q is the pitch on each cycle, using **[N1.151 Peck-drilling withdraw distance]**.

The action is processed based on the command selection to position the hole and drilling axis. The axes and corresponding commands are as follows:

	G83	G87
Tapping axis	Z	X
Positioning axis	X, C	Z, C

3

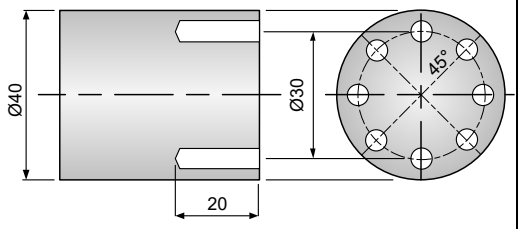
Note:

(1) When the lathe G code is set to A type, the arguments U_, H_ and W_ are for incrementing the values of the X, C and Z axis-direction coordinates.

(2) When the lathe G code is set to B or C type, G90 and G91 are used for switching incremental modes.

(3) In turning and milling mode, if the tapping position is not at absolute coordinate zero (X = 0), the system will return the alarm [0xB6A5: Lathe drilling or tapping instructions are incorrect].

Example description:

Example	Program / Description
	N01 (TAPPING) N02G00 X50.000 N03G00 Z5.000 N04T0101 N05G50 S2000 N06G97 S600 N07M3 N08M8 N09G40 N10G00 X30.000 N11G00 C0.000 N12G99 G84 Z-20.000 P500 Q3 R2.000 L1 F1.500 N13C45. N14C90. N15C135. N16C180. N17C225. N18C270. N19C315. N20G80 N21G00 Z20.000 N22G00 X50.000 N23M09 N24M05 N25M30

3.14.3 Boring cycle (G85/G89)

End face boring cycle		Group	Command code
Lathe	A-Type	09	G85
	B-Type		G85
	C-Type		G85

Side boring cycle		Group	Command code
Lathe	A-Type	09	G89
	B-Type		G89
	C-Type		G89

Instruction format:

G85	X_	C_	Z_	R_	P_	F_	K_
G89	U_	H_	W_				

X_/U_ The G85 instruction is for the X boring coordinate. The G89 instruction is for the hole bottom coordinate.

C_/H_ C axis boring coordinate.

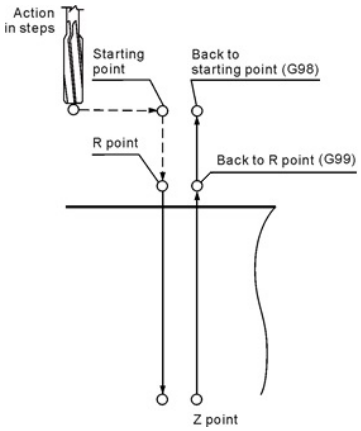
Z_/W_ The G85 instruction is for the hole bottom coordinate. The G89 instruction is for the Z boring coordinate.

R_ Reference point position.

P_ Hole bottom pause time. Unit: s.

F_ Feed rate.

K_ Process repetition.



Instruction description:

The G85 command is usually used with a reamer or boring process and is suitable for high precision hole machining. When the action starts, it is first moved to point R quickly, then it switches to the feed speed of F and cuts to the depth of Z. After the process is finished, the system maintains the same speed to lift the tool and return to point R, and then finally rapidly returns to the starting point. The action is processed based on the command selection to position the hole and drilling axis. The axes and corresponding commands are as follows:

	G83	G87
Boring axis	Z	X
Positioning axis	X, C	Z, C

Note:

(1) When the lathe G code is set to A type, the arguments U_, H_ and W_ are for incrementing the values of the X, C and Z axis-direction coordinates.

(2) When the lathe G code is set to B or C type, G90 and G91 are used for switching incremental modes.

(3) In turning and milling mode, if the tapping position is not at absolute coordinate zero (X = 0), the system will return the alarm [0xB6A5: Lathe drilling or tapping instructions are incorrect].

3.14.4 Canned cycle cancel (G80)

Canned cycle cancel		Group	Command code
Lathe	A-Type	09	G80
	B-Type		G80
	C-Type		G80

Instruction format:

G80

Instruction description:

This instruction is used for canceling the G code group 09 circulation function. It also can operate the G code group 01 to cancel the circulation function, for example G00, G01,G02, G03.

3.15 Turning cycle - Lathe

3.15.1 Finishing cycle (G70/G72)

Finishing cycle		Group	Command code
Lathe	A-Type	09	G70
	B-Type		G70
	C-Type		G72

Instruction format:

G70	P_	Q_

1st block	P First N label number Q Last N label number	

Instruction description:

The finishing cycle command is used as a machining contour turning cycle. This command can be used with the rough turning cycle command (Type A: G71, G72, G73) to specify the P_ to Q_ program label blocks in the common program for final finishing processing.

Note:

If the N label specified by P_ or Q_ cannot be found in the program when executing fine finishing circulation instructions, the system will return the alarm [0x0233 Q_ not found].

3

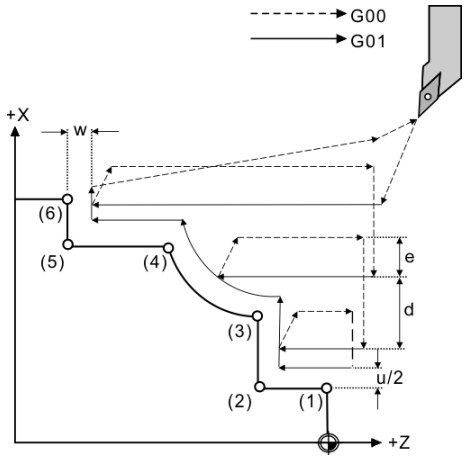
3.15.2 Stock removal cycle in turning (G71/G73)

Stock removal cycle in turning		Group	Command code
Lathe	A-Type	09	G71
	B-Type		G71
	C-Type		G73

Instruction format:

G71	U_	R_	H_				
G71	P_	Q_	U_	W_	F_	S_	T_

1st section	U_ Depth of cut. As shown in d. R_ Escaping amount. As shown in e. H_ Roughing type. 0: Fast escaping mode. 1: Along profile machining mode.
2nd section	P_ Sequence number of the first block for the program of the finishing shape. Q_ Sequence number of the last block for the program of the finishing shape. U_ X axis direction allowance. As shown in u. W_ Z axis direction allowance. As shown in w. F_ Feed rate. T_ Tool number. S_ Spindle speed



Instruction description:

The inner and outer diameter rough turning cycle instructions use P_ and Q_ to set the base for fine turning and to reserve the finishing allowance, and then they use the longitudinal path to quickly remove material. This command is suitable for processing the inner or outer cylindrical materials. The instruction is composed of two consecutive blocks. The first block describes the depth of cut U_, the escaping amount R_ and the roughing type H_; the second block specifies the finishing contour description blocks P_ to Q_ for finishing. The process is based on the X and Z axis to retain the finishing allowance of U_ and W_. The overall roughing path is calculated based on the command settings.

Note:

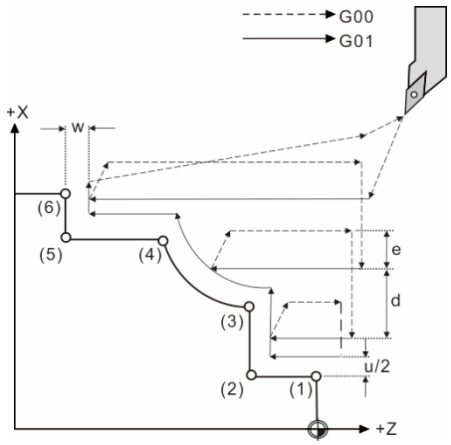
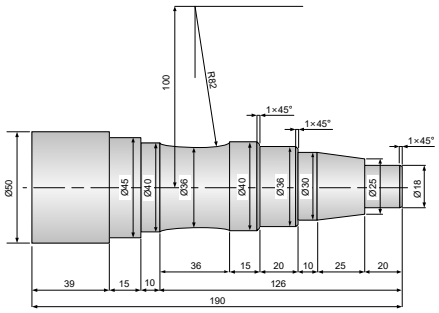
(1) If the finishing allowance U_ is not specified in the first block, then the system will refer to **[N1.170 Cutting depth in G71 / G72 turning cycle]**. The finishing allowance U_ is the relative value, which is not affected by **[N2.001 Bit10 Radius or diameter mode]**.

(2) If the escaping amount R_ is not specified in the first block, the system will refer to **[N1.171 Withdraw amount in G71 / G72 turning cycle]**. The escaping amount R_ is a radius value and is not affected by **[N2.001 Bit10 Radius or diameter mode]**.

(3) If the N label number in the P or Q arguments of the 2nd block does not exist, the system will return the alarm [0x0264 P_Q_ not found].

(4) This instruction uses two continuous blocks as one complete instruction, otherwise the system will return the alarm [0X0247 Multiple loop instructions must have two blocks].

Example description:

Example	Program / Description
	G54 X40.0 Z5.0 M03 S1000 G71 U2. R3. G71 P50 Q60 U0.2 W0.0 F0.2 S1000 N50 G01 X5.0 Z0.0 F0.15 (1) Z-5. (2) X10. (3) G02 X20. Z-15. R10. (4) G01 Z-20. (5) N60 X40. (6) M5 M30
	G00 X70.000 G00 Z20.000 T0101 G98 F1000 G50 S1000 G50 S1000 M3 G00 X65.000 Z10.000 G42 G71 U1.000 R0.5 G71 P210 Q250 U0.300 W0.300 N210 G01 X0.000 Z0.000 G01 X18.000 Z0.000 ,C1.000 G01 X18.000 Z-20.000 G01 X25.000 Z-20.000 G01 X30.000 Z-45.000 G01 X30.000 Z-55.000 G01 X36.000 Z-55.000 ,C1.000 G01 X36.000 Z-75.000 G01 X40.000 Z-75.000 ,C1.000 G01 X40.000 Z-90.000 G02 X40.000 Z-126.000 R80.000 G01 X40.000 Z-136.000 G01 X45.000 Z-136.000 G01 X45.000 Z-151.000 G01 X50.000 Z-151.000 G01 X50.000 Z-151.000 G01 X50.000 Z-190.000 G01 X55.000 Z-190.000 N250 G00 X70.000 G00 Z20.000 M5 M30

3

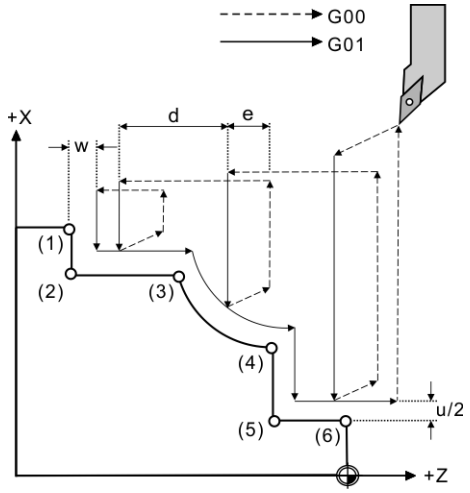
3.15.3 Stock removal cycle in facing (G72/G74)

Stock removal cycle in facing		Group	Command code
Lathe	A-Type	09	G72
	B-Type		G72
	C-Type		G74

Instruction format:

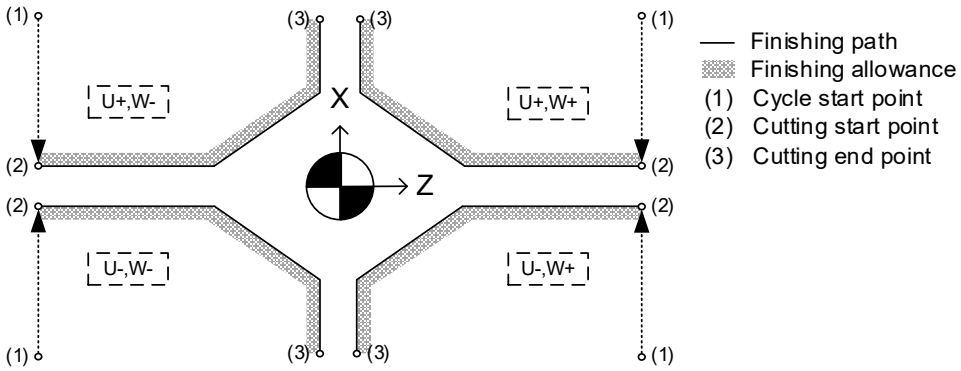
G72	W_	R_	H_				
G72	P_	Q_	U_	W_	F_	S_	T_

1st block	W_ Depth of cut. As shown in d. R_ Escaping amount. As shown in e. H_ Roughing type. 0: Fast escaping mode. 1: Along profile machining mode.
2nd block	P_ Sequence number of the first block for the program of the finishing shape. Q_ Sequence number of the last block for the program of the finishing shape. U_ X axis direction allowance. W_ Z axis direction allowance. F_ Feed rate. T_ Tool number. S_ Spindle speed



Instruction format:

The facing turning cycle instructions use P_ and Q_ to set the base for fine turning and reserve the finishing allowance, and then use the radial path to quickly remove material. This command is suitable for processing the end face of cylindrical materials. The instruction is composed of two consecutive blocks. The first block describes the depth of cut W_, the escaping amount R_ and the roughing type H_; the second block specifies the finishing contour description blocks P_ to Q_ for finishing. The process is based on the X and Z axis to retain the finishing allowance of U_ and W_. The machining direction of this command is related to the finishing reserve amount as shown below.



Note:

(1) If the finishing allowance U_ is not specified in the first block, then the system will refer to

[N1.170 Cutting depth in G71 / G72 turning cycle]. The finishing allowance U_ is the relative value, which is not affected by **[N2.001 Bit10 Radius or diameter mode]**.

(2) If the escaping amount R_ is not specified in the first block, the system will refer to **[N1.171 Withdraw amount in G71 / G72 turning cycle]**. The escaping amount R_ is a radius value, and is not affected by **[N2.001 Bit10 Radius or diameter mode]**.

(3) If the N label number in the P or Q arguments of the 2nd block does not exist, the system will return the alarm [0x0264 P_Q_ not found].

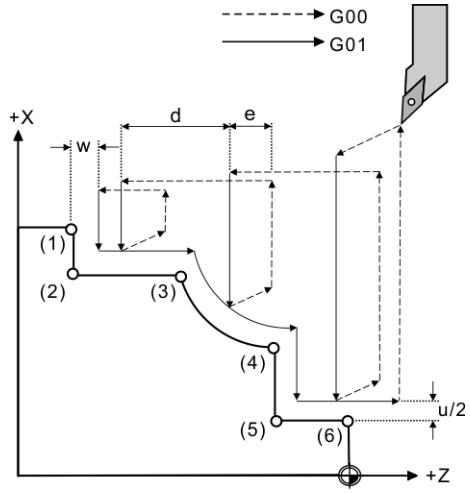
(4) This instruction uses two continuous blocks as one complete instruction, otherwise the system will return the alarm [0X0247 Multiple loop instructions must have two blocks].

(5) In this cycle instruction, the knife nose compensation function is invalid.

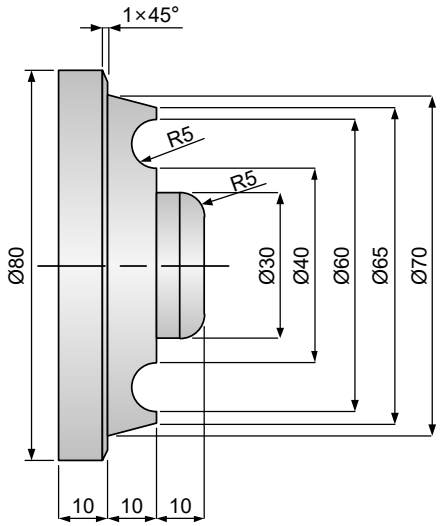
(6) The cycle instruction will be ignored if the single block does not contain a movement command or the N, F, S, M, T arguments.

(7) The G72 instruction is not a continuous instruction.

Example description:

Example	Program / Description
Example 1: 	G54 X40.0 Z5.0 M03 S1000 G01 X45. Z5. F0.2 G72 W2. R3. G72 P50 Q60 U0.2 W0.0 F0.15 N50 G01 X40. Z-20. (1) X30. (2) Z-15. (3) G03 X20. Z-5. R10. (4) G01 X5. (5) N60 Z0. (6) M5 M30

3

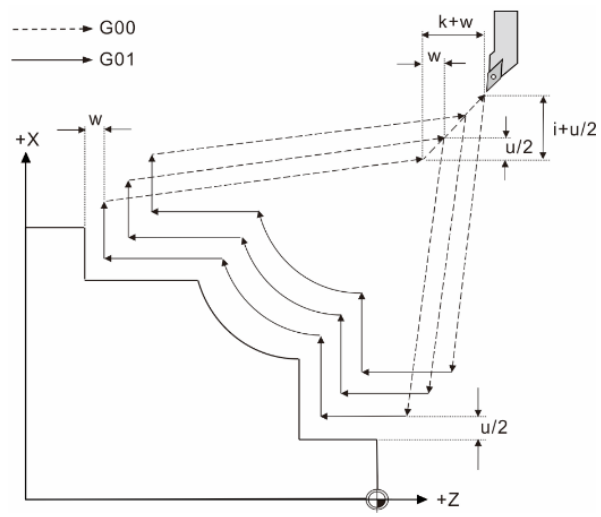
Example	Program / Description
Example 2:  	G00 X90.000 G00 Z20.000 T0101 G99 F0.500 G50 S2000 G97 S1000 M3 G00 X85.000 Z10.000 G40 G72 W2.000 R0.5 G72 P210 Q250 U0.500 W0.500 F0.500 N210 G01 X80.000 Z-30.000 F0.200 G01 X80.000 Z-20.000 ,C1.000 G01 X70.000 G01 X65.000 Z-10.000 G01 X60.000 G03 X40.000 R5.000 G01 X30.000 G01 Z0.000 ,R5.000 G01 X0.000 Z0.000 N250 G00 X90.000 G00 Z20.000 M09 M05 M30

3.15.4 Profile rough turning cycle (G73/G75)

Profile rough turning cycle		Group	Command code
Lathe	A-Type	09	G73
	B-Type		G73
	C-Type		G75

Instruction format:

G73	U_	W_	R_	H_			
G73	P_	Q_	U_	W_	F_	S_	T_

1st block	U_ Total distance to cut in X axis direction. W_ Total distance to cut in Z axis direction. R_ Cutting times. H_ Roughing type. 0: Fast escaping mode. 1: Along profile machining mode.	
2nd block	P_ Sequence number of the first block for the program of the finishing shape. Q_ Sequence number of the last block for the program of the finishing shape. U_ X axis direction allowance. As shown in u. W_ Z axis direction allowance. As shown in w. F_ Feed speed T_ Tool number. S_ Spindle speed	

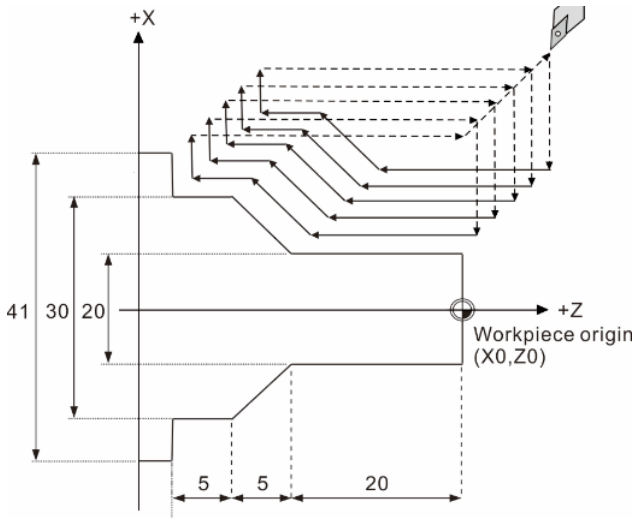
Instruction description:

The profile rough turning cycle instructions use P_ and Q_ to set the base path for fine turning and define the total distance to cut in U_ and W_, with R_ being the number of times to cut. This command is often used in the reprocessing processes of rough workpieces such as casting, forging, etc. The instruction is composed of two consecutive blocks. The first block describes the total distance to cut in U_ and W_ and R_ as the number of times to cut; the second block specifies the finishing contour description blocks P_ to Q_ for finishing. The process is based on the X and Z axis to retain the finishing allowance of U_ and W_. The single pitch is the total removal amount of each axis divided by the number of cuts (R-1).

X axis single feeding amount $\mathrm{u} = \frac{U}{R-1}$, Z axis single feeding amount $\mathrm{w} = \frac{W}{R-1}$

3

Example description:

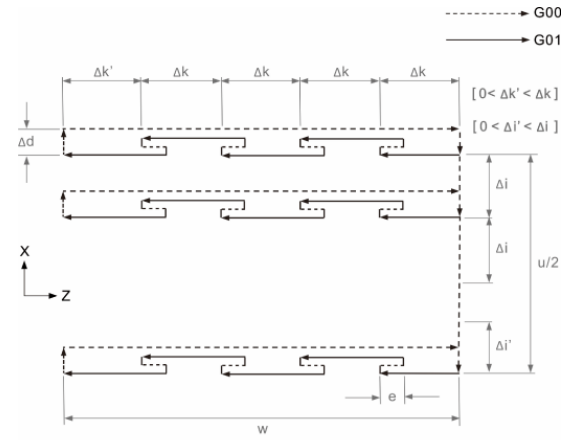
Example	Program / Description
	T0101 M03 S1600 G00 X41.000 Z2.000 G73 U10.000 W10.000 R5.000 (X,Z total removal 10mm, cycle times 5 G73 P50 Q60 U0.400 W0.200 F0.250 N50 G00 X20.000 G01 Z-20.000 F0.12 G01 X30.000 Z-25.000 G01 W-5.000 N60 X41.000

3.15.5 End face peck drilling cycle (G74/G76)

End face peck drilling cycle		Group	Command code
Lathe	A-Type	09	G74
	B-Type		G74
	C-Type		G76

Instruction format:

G74	R_					
G74	X_	Z_	P_	Q_	R_	F_
	U_	W_				

1st block	R_ Tool withdraws in Z axis direction.	
2nd block	X_ Target position of X axis. U_ Incremental distance of X axis. Z_ Target position of Z axis. W_ Incremental distance of Z axis. P_ Pitch of X axis. unit: um Q_ Pitch of Z axis. Unit: um. R_ Bottom escape distance of X axis. F_ Feed speed	

Instruction description:

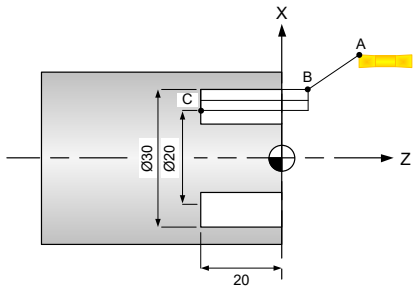
The G74 command is mainly used in end face groove processing. The cycle is automatically executed in the end face direction according to the turning target coordinates, cutting pitch, tool offset, bottom escape distance and other argument values in the command. △k is the Z axis turning pitch in each cycle, and then the value of e is the withdraw distance. Turning continues until the bottom target of the Z axis position is reached. After reaching the bottom, the tool escape amount is △d, and then it returns rapidly to the starting point of the Z axis. Then the tool moves for a distance of △i in the X axis direction and continues the above action until it reaches the target of the X axis position.

Note:

Do not use the tool radius compensation simultaneously.

3

Example description:

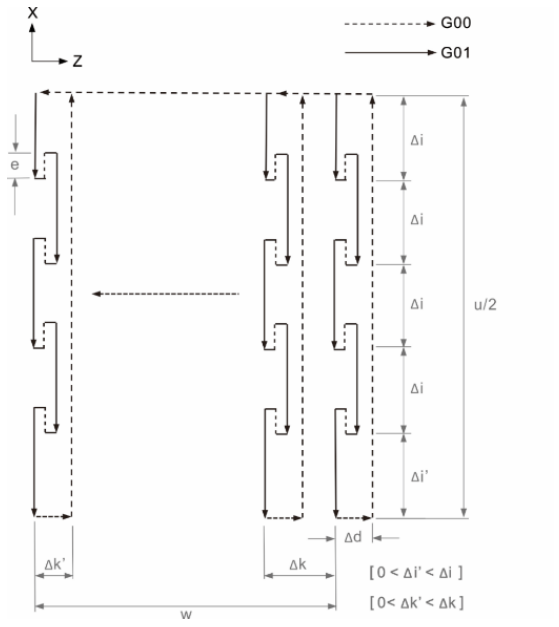
Example	Program / Description
	G00Z20.000 G00X50.000 (A: X50.0, Z20.0) T0101 G99 F0.500 (Feed rate setting G50 S1500 (Spindle Max. speed setting G97 S1000 (Spindle speed setting M03 M08 G40 (Disable tool compensation G00 X30.000 Z5.000 G01 Z5.000 (Cycle initial position (B:X30.000, Z5.000) G74 R0.500 (Withdraw in Z axis direction (X axis feeding 2.5mm, Z axis peck feeding 2.0mm (Cycle target position C (X20.0, Z-20.0) G74 X20.000 Z-20.000 P2500 Q2000 R0.000 G01 Z5.000 G00 Z5.000 G00 Z20.000 G00 X50.000 M09 M05 M30

3.15.6 Outer/internal diameter drilling cycle (G75/G77)

Outer/internal diameter drilling cycle		Group	Command code
Lathe	A-Type	09	G75
	B-Type		G75
	C-Type		G77

Instruction format:

G75	R_					
G75	X_	Z_	P_	Q_	R_	F_
	U_	W_				

1st block	R_ Tool withdraws in X axis direction.	
2nd block	X_ Target position of X axis. U_ Incremental distance of X axis. Z_ Target position of Z axis. W_ Incremental distance of Z axis. P_ Pitch of X axis. unit: um Q_ Pitch of Z axis. Unit: um. R_ Bottom escape distance of Z axis. F_ Feed speed	

Instruction description:

The G75 command is mainly used in end face groove processing. The cycle is automatically executed in the end face direction according to the turning target coordinates, cutting pitch, tool offset, bottom escape distance and other argument values in the command. △i is the X axis turning pitch in each cycle, and then the value of e is the withdraw distance. Turning continues until the bottom target of the X axis position is reached. After reaching the bottom, the tool escape amount is △d, and then it returns rapidly to the starting point of the X axis.

Then the tool moves for a distance of △k in the Z axis direction and continues the above action until it reaches the target of the Z axis position.

Note:

G75 instruction cannot use tool nose compensation simultaneously.

3

3.15.7 Outer/internal diameter cutting cycle (G90/G77/G20)

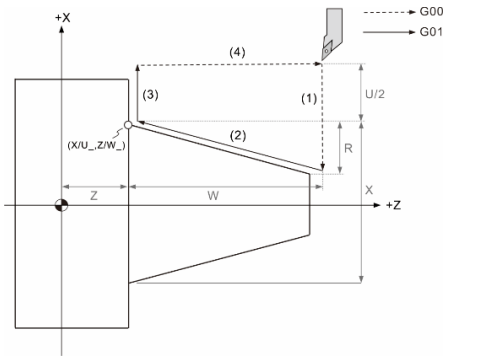
Outer/internal diameter cutting cycle		Group	Command code
Lathe	A-Type	09	G90
	B-Type		G77
	C-Type		G20

Instruction format:

G90	X_	Z_	R_	F_
	U_	W_		

1st block

X_ Target position of X axis.
U_ Incremental distance of X axis.
Z_ Target position of Z axis.
W_ Incremental distance of Z axis.
R_ Taper value.
F_ Feed speed.



Instruction description:

The G90 single type turning cycle command will execute a complete linear (taper) turning cycle of the outer diameter of the workpiece.

The G90 command action steps are as follows:

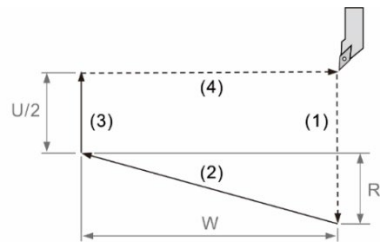
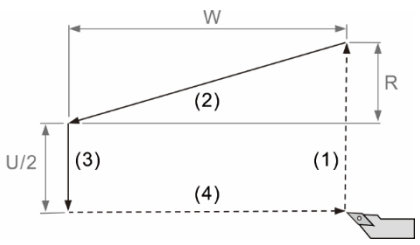
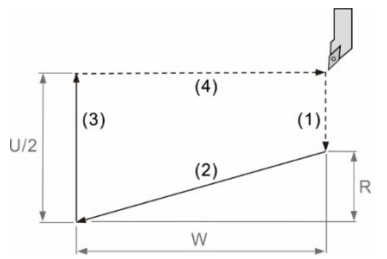
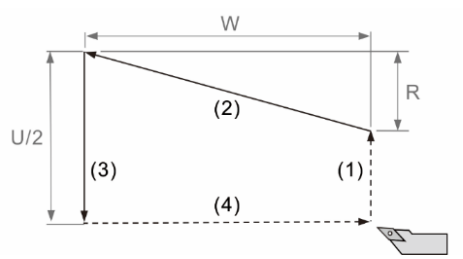
(1) Beginning from the starting point, the tool rapidly traverses the X axis to the specified position plus the taper value.

(2) Linear cutting to the specified coordinates of the Z axis and X axis.

(3) Linear cutting to the starting position on the X axis.

(4) Rapidly move back to the starting position.

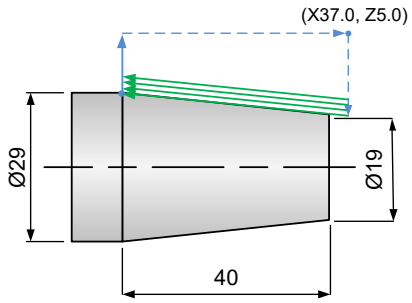
U, W, R corresponding turning direction:

Outer turning	Inner turning
1. U < 0, W < 0, R < 0	3. U > 0, W < 0, R < 0 at |R|≤|U/2|
	
2. U < 0, W < 0, R > 0 at |R|≤|U/2|	4. U > 0, W < 0, R > 0
	

Note:

The G90 single type turning cycle is a continuously valid command. To cancel the G90 command, users need to use the G92 or G94 commands, the G80 canned cycle cancel function, or other G code group 1 commands such as G00, G01, G02 or G03.

Example description:

Example	Program / Description	
	Main program	
	G00 X50.000	
	G00 Z20.000	
	T0303	
	G99 F0.500	
	G97 S2000	
	M3	
	M8	
	G40	
	G00 X37.000 Z5.000	
	G90 X35.000 Z-40.000 R-5.000	End of cycle turning (1)
	X33.000	End of cycle turning (2)
	X31.000	End of cycle turning (3)
	X29.000	End of cycle turning (4)
	G00 X50.000	
	G00 Z20.000	
	M09	
	M05	
	M30	

3

3.15.8 End face turning cycle (G94/G79/G24)

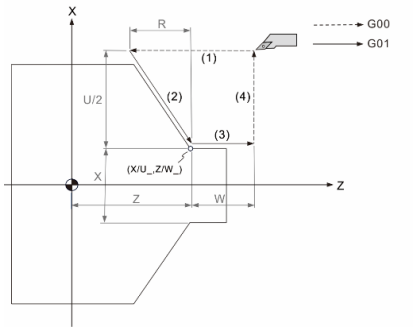
End face turning cycle		Group	Command code
Lathe	A-Type	09	G94
	B-Type		G79
	C-Type		G24

Instruction format:

G94	X_	Z_	R_	F_
	U_	W_		

1st block

X_ Target position of X axis.
U_ Incremental distance of X axis.
Z_ Target position of Z axis.
W_ Incremental distance of Z axis.
R_ Taper value.
F_ Feed speed.



Instruction description:

The G94 single type turning cycle command will execute a complete linear (taper) turning cycle of the end face of the workpiece.

The G94 command action steps are as follows:

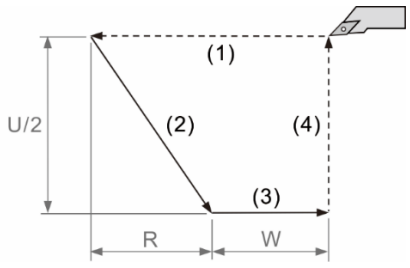
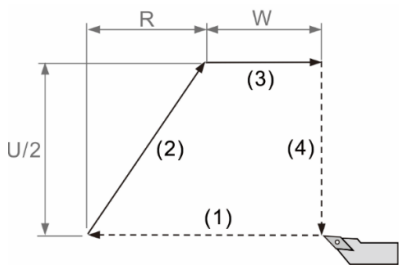
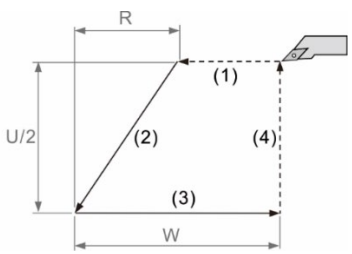
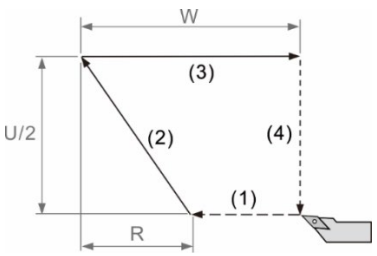
(1) Beginning from the starting point, the tool rapidly traverses the Z axis to the specified position plus the taper value.

(2) Linear cutting to the specified coordinates of the Z axis and X axis.

(3) Linear cutting to the starting position on the Z axis.

(4) Rapidly move back to the starting position.

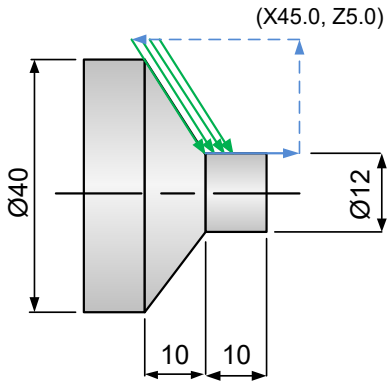
U, W, R corresponding to the turning direction:

Outer turning	Inner turning
1. U < 0, W < 0, R < 0	3. U > 0, W < 0, R < 0 at |R|≤|W|
	
2. U < 0, W < 0, R > 0 at |R|≤|W|	4. U > 0, W < 0, R > 0
	

Note:

The G94 single type end face turning cycle is a continuously valid command. To cancel the G94 command, users need to use other cycle commands such as the G90 or G92, the G80 canned cycle cancel function, or other G code group 1 commands such as G00, G01, G02 or G03.

Example description:

Example	Program/ Description
	G00 X50.000 G00 Z20.000 T0101 G99 F0.500 G97 S2000 M3 M8 G40 G00X45.0 Z5.0 G90X12.0 Z-7.0R-5. (End of cycle turning 1) Z-8.000 (End of cycle turning 2) Z-9.000 (End of cycle turning 3) Z-10.000 (End of cycle turning 4) G00 X50.000 G00 Z20.000 M09 M05 M30

3.16 Threading cycle - Lathe

3

3.16.1 Threading (G32/G33)

Threading		Group	Command code
Lathe	A-Type	01	G32
	B-Type		G33
	C-Type		G33

Instruction format:

G32	X_	Z_	F_	Q_

X/U_ Target position of threading.

F_ Thread leads; the linear distance of one thread rotation.

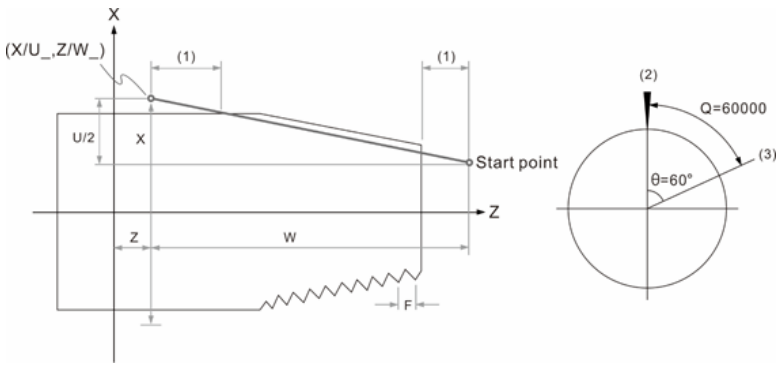
Q_ Thread starting offset angle.

The unit is 0.001 degrees. Default as 0 degrees when it is not specified.

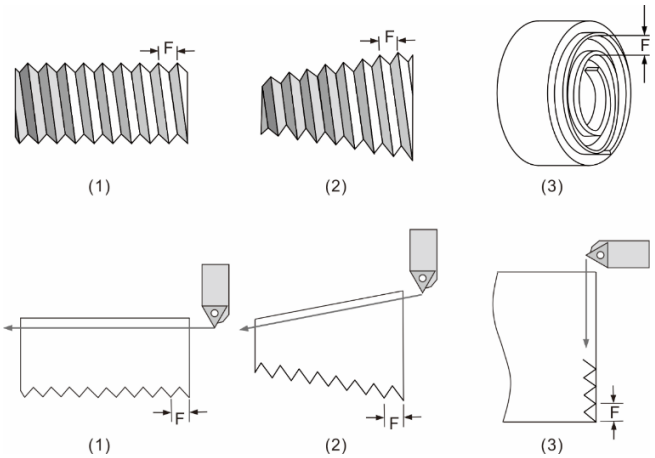
Instruction description:

The threading command specifies the thread target position with X_/U_ and Z_/W_. Thread processing such as linear thread, taper thread, and spiral thread is established according to the relationship between the starting point and the end point; among them, the thread pitch (lead) is determined by the argument F_.

In the figure below, the label (2) indicates the spindle Z-phase signal. The argument Q_ specifies the position of the thread entry point labeled (3). Its unit is 0.001 degrees, and it is limited to integers.



Thread types:



(1) Linear thread. (2) Taper thread. (3) Surface screw thread.

Note:

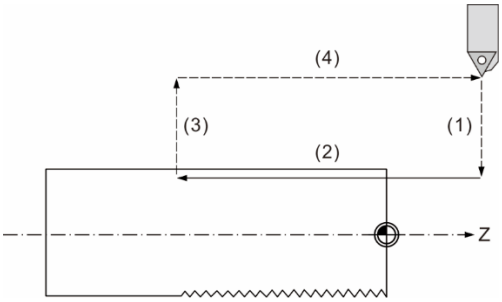
(1) The threading must be operated at a fixed spindle speed and the spindle ratio should be kept at 100% speed.

(2) When pressing the program stop during threading, the process will not stop immediately, but will stop at the end point of the next non-threading block. However, pressing RESET will immediately stop threading and may cause thread damage.

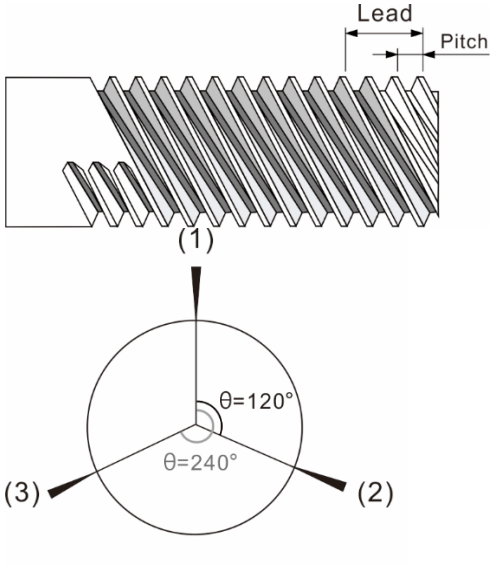
(3) When the feed axis exceeds the maximum speed limit, an alarm [0xB01D Spindle speed is over speed] will be issued, and the spindle speed should be reduced. For example: The spindle speed is 3000 RPM, and the thread pitch (F) is 1.5, then the Z axis feed speed when threading will be 4500 mm/min.

(4) Due to the forward and retract action of thread processing, some invalid threads will be produced at the starting point and end point of the thread. This phenomenon will affect the use of the thread function. It is recommended to extend the thread processing length when thread turning to make it slightly longer than actual requirements to avoid this.

Example description:

Example	Program / Description	
Single threading:  	T0101	
	M03 S1000	
	G00 X40.000 Z15.000	
	X17.450	(1) 1st thread depth.
	G32 Z-30.000 F1.5	(2) Threading.
	G00 X40.000	(3) X axis fast withdraw.
	G00 Z15.000	(4) Z axis fast withdraw.
	G00 X17.200	(1) 2nd thread depth.
	G32 Z-30.000 F1.5	(2) Threading.
	G00 X40.000	(3) X axis fast withdraw.
	G00 Z15.000	(4) Z axis fast withdraw.
	X17.000	(1) 3rd thread depth.
	G32 Z-30.000 F1.5	(2) Threading.
	G00 X40.000	(3) X axis fast withdraw.
	G00 Z15.000	(4) Z axis fast ithdraw.
	G00 X16.850	(1) 4th thread depth.
	G32 Z-30.000 F1.5	(2) threading.
	G00 X40.000	(3) X axis fast withdraw.
	G00 Z15.000	(4) Z axis fast withdraw.
	G00 X16.800	(1) 5th thread depth.
	G32 Z-30.000 F1.5	(2) Threading.
	G00 X40.000	(3) X axis fast withdraw.
	G00 Z15.000	(4) Z axis fast withdraw.
	M05	
	M30	

3

Example	Program / Description
Multi-threading: 	L(lead) = n (No. of thread) x pitch (thread pitch) Main program T0202 M03 S1000 G00 X45.000 G00 Z10.000 G66 P0001 A0 Thread offset angle (1) X17.45 X17.20 X17.00 G67 G66 P0001 A120000 Thread offset angle (2) X17.45 X17.20 X17.00 G67 G66 P0001 A240000 Thread offset angle (3) X17.45 X17.20 X17.00 G67 G00 X45.000 G00 Z10.000 M30 (Sub-program) O0001 (G66 P0001 A_ argument value inserted into the subprogram #1) (#1 means thread offset angle) G32 Z-30.000 F3.000 Q#1 G00 X45.000 G00 Z10.000 M99

3.16.2 Threading cycle (G92/G78/G21)

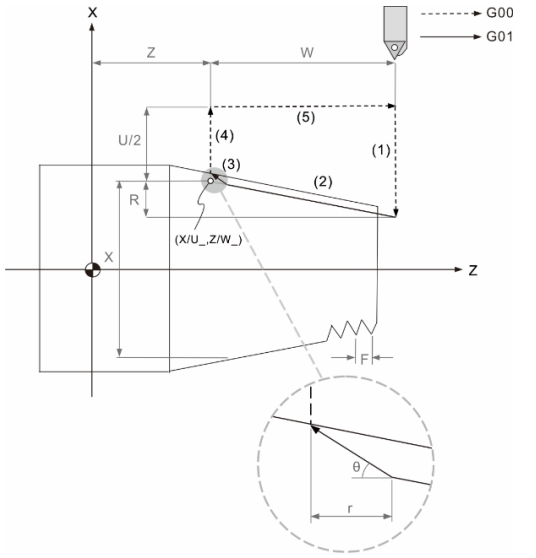
Threading cycle		Group	Command code
Lathe	A-Type	09	G92
	B-Type		G78
	C-Type		G21

Instruction format:

G92	X_	Z_	R_	F_	Q_
	U_	W_			

1st block

X_/ U_	Target position of threading.
Z_/ W_	Target position of threading.
R_	Taper value. If not specified the system will execute parallel threading.
F_	Thread lead.
Q_	Thread offset angle. Unit: 0.001 degrees.



Instruction description:

A single thread turning cycle command is used to turn a complete outer diameter thread of the workpiece in a single cycle. It can be a parallel threading or tapered threading cycle according to the taper value setting. The threading chamfering angle (θ in the figure) and the threading chamfer length setting value (r in the figure) are set by parameters **[N1.177] and [N1.178]** respectively.

The G92 command action steps are as follows:

(1) Beginning from the starting point, the tool rapidly traverses the X axis to the specified position plus the taper value.

(2) Threading to the specified coordinates of the Z axis and X axis.

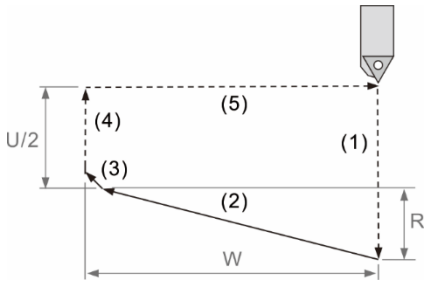
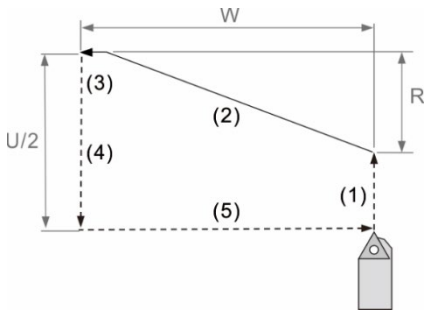
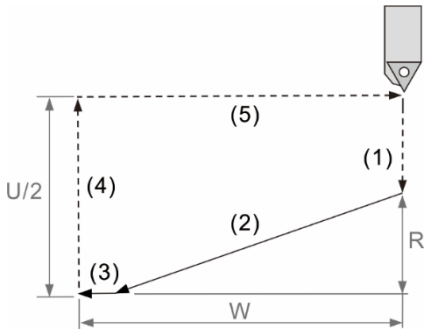
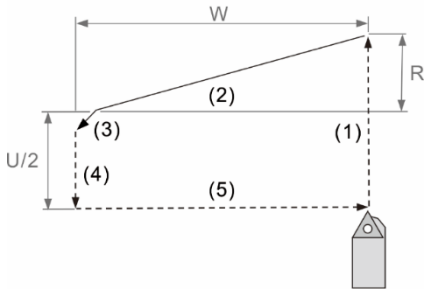
(3) Perform chamfer command.

(4) Rapidly move on the X axis to the starting position.

(5) Rapidly move back to the starting position.

3

U, W, R corresponding turning direction:

Outer turning	Inner turning
1. U < 0, W < 0, R < 0	3. U > 0, W < 0, R < 0 at |R|≤|U/2|
	
2. U < 0, W < 0, R > 0 at |R|≤|U/2|	4. U > 0, W < 0, R > 0
	

Notes:

(1) The G92 single thread turning cycle is a continuously valid command. To cancel the G92 command, users need to use other cycle commands such as the G90 or G94, the G80 canned cycle cancel function, or other G code group 1 commands such as G00, G01, G02 or G03.

(2) Parameter **[N1.177]** defines the chamfer angle in G76 / G92 threading.
Parameter **[N1.178]** defines the chamfer length in G76 / G92 threading.

(3) For key points to note in thread turning, please refer to the G33 thread turning instructions.

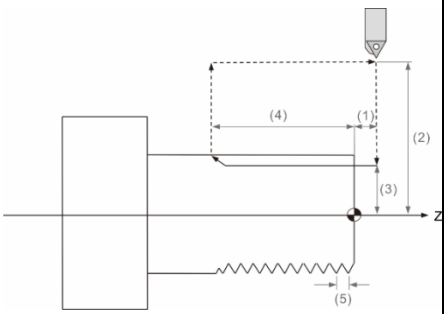
(4) The threading must be operated at a fixed spindle speed and the spindle ratio should be kept at 100% speed.

(5) When pressing the program stop during threading, the process will not stop immediately, but will stop at the end point of the next non-threading block. However, pressing RESET will immediately stop threading and may cause thread damage.

(6) When the feed axis exceeds the maximum speed limit, an alarm [0xB01D Spindle speed is over speed] will be issued, and the spindle speed should be reduced. For example: The spindle speed is 3000 RPM, and the thread pitch (F) is 1.5, then the Z axis feed speed when threading will be 4500 mm/min.

(7) Due to the forward and retract action of thread processing, some invalid threads will be produced at the starting point and end point of the thread. This phenomenon will affect the use of the thread function. It is recommended to extend the thread processing length when thread turning to make it slightly longer than actual requirements to avoid this.

Example description:

Example	Program / Description	
Single thread turning circulation:  	T0101	(Select tool 1 compensation)
	M03 S2000	(Spindle positive rotation 2000 RPM)
	G00 Z5.000	(1) Z axis threading initial point.
	X30.000	(2) X axis withdraw distance.
		(3) thread depth 17.65 mm.
		(4) length 25 mm.
		(5) thread lead value 1.5 mm.
	G92 X17.65 Z-25.0 F1.5	
	X17.45	(Thread depth)
	X17.25	(Thread depth)
	X17.05	(Thread depth)
	X16.85	(Thread depth)
	X16.65	(Thread depth)
	X16.45	(Thread depth)
	X16.25	(Thread depth)
	X16.05	(Thread depth)
	X15.9	(Thread depth)
	M05	(Spindle stop)
	G00 X50.000	(Tool return to a safe distance)
	G00 Z10.000	(Tool return to a safe distance)
	M30	(Program finish)

3

3.16.3 Multiple-thread cutting cycle (G76/G78)

Multiple-thread cutting cycle		Group	Command code
Lathe	A-Type	09	G76
	B-Type		G76
	C-Type		G78

Instruction format:

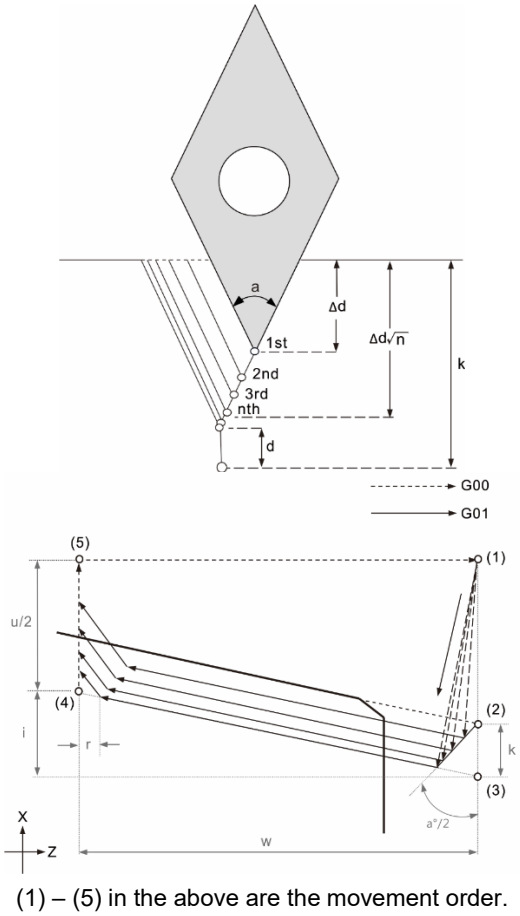
G76	P_	Q_	R_				
G76	X_	Z_	P_	Q_	R_	L_	F_
	U_	W_					

1st block

P_		Composed of three groups with 2-digit values: (m), (r), and (a).
	m	Fine turning times. Range: 01 ~ 99.
	r	Chamfer angle. Range: 00 ~ 99.
	a	Tool top angle. Selection: 00, 29, 30, 55, 60, 80.
Q_		Minimum cutting depth.
R_		Fine turning allowance.

2nd block

X_/U_	Threading target position.
Z_/W_	Threading target position.
P_	Thread height.
Q_	Thread initial offset angle. unit: 0.001 degree
R_	Thread radius difference.
L_	Selection of tool direction in threading cycle.
	Not specified: cut on the right side.
	L=0: cut to the right side.
	L=1: cut to the center
	L=2: cut to the left side
	L=3: cut direction left and right staggered
F_	thread lead
K_	thread pitch per rotation



(1) – (5) in the above are the movement order.

Instruction description:

In the compound thread turning cycle, the force on the cutting-edge feed is exerted on one side to facilitate rapid processing requirements. After the target position is given in the thread turning cycle, the threading cycle will be performed with a fixed cutting amount according to the given number of cuts.

Notes:

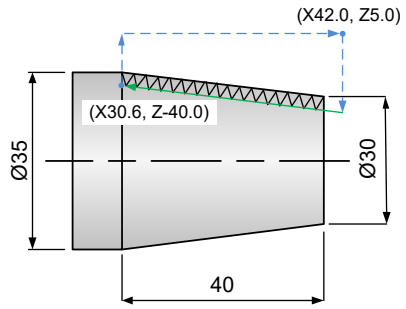
(1) The threading must be operated at a fixed spindle speed and the spindle ratio should be kept at 100% speed.

(2) When pressing the program stop during threading, the process will not stop immediately, but will stop at the end point of the next non-threading block. However, pressing RESET will immediately stop threading and may cause thread damage.

(3) When the feed axis exceeds the maximum speed limit, an alarm [0xB01D Spindle speed is over speed] will be issued, and the spindle speed should be reduced. For example: The spindle speed is 3000 RPM, and the thread pitch (F) is 1.5, then the Z axis feed speed when threading will be 4500 mm/min.

(4) Due to the forward and retract action of thread processing, some invalid threads will be produced at the starting point and end point of the thread. This phenomenon will affect the use of the thread function. It is recommended to extend the thread processing length when thread turning to make it slightly longer than actual requirements to avoid this.

Example description:

Example	Program / Description
	Instruction description: P(m)(r)(a) = P10160 m=1, fine turning times; r=01, the amount of thread chamfer angle; a=60, threading angle. Q(△dmin)=Q200, Minimum cutting depth 0.2mm; R0.1= Fine allowance value 0.1mm (X30.600, Z-40.000) = thread target position. R(i)=R-5.000, thread radius difference (thread inclination) P(k)=P2200, thread height 2.2mm (specified radius) Q(△d)=Q500, 1st cutting depth 0.5mm L0=thread feeding direction to the right Main program: G00 X50.000 G00 Z20.000 T0303 G99 F0.500 G97 S2000 M3 M8 G40 G00 X37.000 Z5.000 G76 P10160 Q200 R0.100 G76 X30.600 Z-40.000 R-5. P2200 Q500 L0 F2.000 G00 X50.000 G00 Z20.000 M09 M05

3

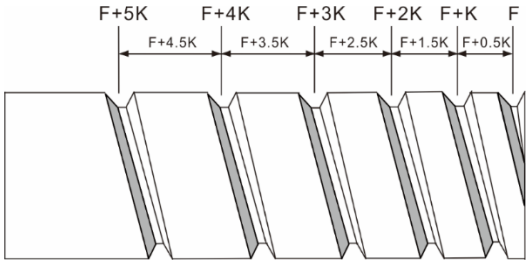
3.16.4 Variable lead threading (G34)

Variable lead threading		Group	Command code
Lathe	A-Type	01	G34
	B-Type		G34
	C-Type		G34

Instruction format:

G34	X_	Z_	F_	Q_
	U_	W_		

1st block	X_/U_	Target position of threading.
	Z_/W_	Target position of threading.
	Q_	Thread initial offset angle. unit: 0.001 degree
	F_	thread lead
	K_	thread pitch per rotation



Instruction description:

G34 variable lead thread cutting completes variable lead thread cutting according to the specified thread lead increase or decrease.

Lead: The linear distance of each revolution of the threading.

Pitch: The distance between two threads.

Lead distance: $\mathrm{d} = \mathrm{Vo\,t} + \frac{1}{2}\mathrm{a}t^2$

Vo: Thread initial pitch (F)

t: Thread turn number.

a: Pitch increase value per rotation (K)

Example description:

1st circle lead distance $\mathrm{d} = \mathrm{F} * 1 + \frac{1}{2} * K * 1^2 = F + 0.5K$

Lead distance between 1st and 2nd circle $\mathrm{d} = \mathrm{F} * 2 + \frac{1}{2} * K * 2^2 = 2F + 2K$

This lead value is the total of the 1st and 2nd circle distances, and the second lead distance is obtained by subtracting the first lead distance.

$$(2\mathrm{F} + 2\mathrm{K}) - (\mathrm{F} + 0.5\mathrm{K}) = \mathrm{F} + 1.5\mathrm{K}$$

Notes:

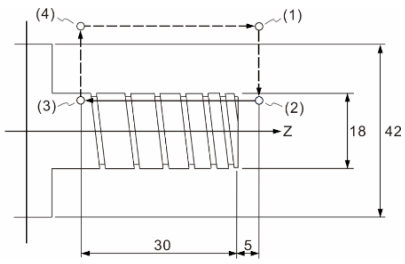
(1) The threading must be operated at a fixed spindle speed and the spindle ratio should be kept at 100% speed.

(2) When pressing the program stop during threading, the process will not stop immediately, but will stop at the end point of the next non-threading block. However, pressing RESET will immediately stop threading and may cause thread damage.

(3) When the feed axis exceeds the maximum speed limit, an alarm [0xB01D Spindle speed is over speed] will be issued, and the spindle speed should be reduced. For example: The spindle speed is 3000 RPM, and the thread pitch (F) is 1.5, then the Z axis feed speed when threading will be 4500 mm/min.

(4) Due to the forward and retract action of thread processing, some invalid threads will be produced at the starting point and end point of the thread. This phenomenon will affect the use of the thread function. It is recommended to extend the thread processing length when thread turning to make it slightly longer than actual requirements to avoid this.

Example description:

Example	Program /Description
	(Main program) T0101 (Select tool 1) M03 S600 (Spindle positive rotate 600 RPM) G00 X50.000 Z5.000 (Rapidly move to engage point) G66 P0001 L1 (Macro call instruction: (Execute once sub-program O0001 X17.65 (Thread depth) X17.45 (Thread depth) X17.30 (Thread depth) X17.20 (Thread depth) X17.10 (Thread depth) X17.05 (Thread depth) G67 (Macro call instruction finish) G00 X50.000 Z5.000 (Rapidly move to tool withdraw) M05 (Spindle stop) M30 (Program finish) (Sub-program) O0001 (Thread length Z-30.) (Thread pitch increase 0.5 mm/rev, initial thread pitch 1 mm) G34 Z-30.000 K0.5 F1.000 G00 X50.000 G00 Z5.000 M99 (Return to main program)

CNC Alarm Information A

This appendix provides the information about the alarms and troubleshooting methods for the NC system. Search this appendix for the methods of handling the NC system related malfunctions.

A.1 CNC Alarm Categories

A

The CNC alarms can be divided into three categories, which are system alarms, user-defined alarms, and user-defined macro alarms. This appendix only describes the **[system alarms]** while the rest are user-defined.

Alarm Category	Alarm Code	Alarm Description
System alarms	-	The system alarms caused by system error or operation error. Some system alarms include information, which has different meanings according to the different alarms.
User-defined alarms	A _	The user-defined alarms which are programed in the MLC. When the A_ device is triggered, the alarm corresponding to the A_ device defined in **[DOPSoft - User-defined alarm]** will reported as well. Range: A0~A511
User-defined macro alarms	MR_	The user-defined macro alarm works with NC program # variable. When the command as #20020=_, the alarm triggered and have the corresponding macro alarm defined in **[DOPSoft - Macro-defined alarm]** will displayed on the controller. Range: M1~M1000

A.2 CNC System Alarms

The system alarms are divided into MLC related alarms and NC related alarms by function.

System alarm category	Alarm code range	Abnormal action	Description
NC alarms	0x0000 ~ 0x1FFF	NC error	The alarms in this range are the error code that terminate the execution when an error occurs during the operation of the NC system, mainly divided into system abnormalities or operation error alarms. If the reason of the error cannot be identified, please report to Delta for assistance.
MLC alarms	0xA000 ~ 0xAFFF	MLC error	The alarms in this range are the warning errors reported when the MLC and HMI screen inside the controller make errors during operation. If there is a related alarm, powering on the CNC again first, and if it still cannot be resolved, please contact Delta or supplier for assistance.
HMI alarms	0x8000 ~ 0x8FFF	HMI error	
Servo related alarms	0xF002	-	The error code is an alarm when the errors of servo axes occur during system operation. When the error occurs, the servo alarm message will display on the screen.

A.2.1 NC Instruction Errors

Alarm code (Hex)	Name	Cause and correction
0x0000	NC internal error.	An internal error occurred in the control system, please contact Delta or supplier.
0x0002	The specified line number or N line number cannot be found.	The specified file message cannot be found.
0x0004	Macro file call error.	Macro number error.
0x0005	Macro call mode error.	System mode error.
0x0201	Undefined operator.	Incorrect syntax. (Undefined symbol) Wrong example: G01O100.; “O” is undefined.
0x0202	Repeated operators command in one line.	Incorrect syntax. Please correct the syntax. Wrong example: G01 X100 G01 X50.
0x0203	Operators that must be command at the beginning of the block are not written at the beginning of the line.	Specific commands need to be used at the beginning of the line. Wrong example: G01 IF ➔ IF misused. G01 WHILE ➔ WHILE is misused.
0x0204	Operators that are forbidden to be command at the beginning of the block are written at the beginning of the line.	Specific commands cannot be used at the beginning of the line. Ex: DO, THEN
0x0205	Operators that must be command at the end of the block are not written at the end of the line.	Specific commands are not used at the end of the line. Wrong example: END 1 X100. ➔ END misused.
0x0206	Operators’ syntax error.	Wrong example: #100=*3 ➔ character * misused.
0x0207	Command syntax error.	Wrong example: X10. G1 ➔ command “X10.” misused.
0x0208	Command syntax error. Statements and motion command cannot program in the same line.	Wrong example: #1=2 G01 X10. ➔”#1=2” and “G01 X10.” cannot be on the same line.
0x0209	Command syntax error. Wrong use of brackets.	“[“ and ”]” are not paired.
0x020A	Command syntax error. Wrong command characters.	Example error: #1=BIT[2,SIN[3]]. The command content is incorrect with a comma. Only numbers or ‘#’ for arithmetic operations are allowed.
0x020B	Command syntax error. Cannot resolve multi-input operators.	Multi-input instructions such as ATAN can’t recognized by the system when it comes to negative numbers or the minus sign.
0x020C	Command syntax error. Newline character error.	Newline character incomplete.
0x020D	Command syntax error. Violation of judgement operator in command syntax.	Syntax error. Correct the syntax format. Invalid syntax for judgement operators such as ‘>’, ‘<’, ‘<>’, ‘=’. Error example: #1>G01.
0x020E	Command syntax error. IF command error.	Syntax error. Correct the syntax format. There is no condition after the IF judgement.
0x020F	Command syntax error. WHILE condition statement error.	Syntax error. Correct the syntax format. There is no condition after the WHILE statement.

A

Alarm code (Hex)	Name	Cause and correction
0x0210	Command syntax error. Divisor is 0.	Syntax error. Correct the syntax format. Expression divided by zero, unable to compute.
0x0211	Command syntax error. Assignment operator used incorrectly.	Syntax error. Correct the syntax format. Incorrect syntax for the '=' operator. Error example: #1=G01.
0x0212	The macro program is being called more times than the allowed limit.	Subprogram or macro program stack exceeds 8 levels.
0x0213	Command syntax error. WHILE loop command without END instruction.	Syntax error. Correct the syntax format. The WHILE statement lacks the END instruction.
0x0214	Command syntax error. The loop count for DO is not an integer.	The character following DO can only be an integer.
0x0215	Command syntax error. The WHILE command lacks a DO instruction in its syntax.	Missing DO instruction in the syntax of the WHILE command.
0x0216	Command syntax error. The relevant instruction is missing the D parameter.	Missing usage of the D parameter in the command syntax. Error example: G42 X100.
0x0217	Command syntax error. The value of D exceeds the maximum limit.	The value of D parameter exceeds the maximum limit. Error example: G42 D1234567.
0x0218	Command syntax error. The relevant instruction is missing the H parameter.	Missing usage of the H parameter in the command syntax. Error example: G43 X100.
0x0219	Command syntax error. The value of H exceeds the maximum limit.	The value of H parameter exceeds the maximum limit. Error example: G43 H1234567.
0x021A	The T value of lathe tool compensation exceeds the maximum limit.	Error example: M06 T1234567.
0x021B	Error in using the G53.1 command.	The G53.1 command must be used after the G68.2 command.
0x021C	The T value exceeds the maximum limit.	Error example: M06 T1234567.
0x021D	Not supported G code instruction.	Syntax error. Correct the syntax format. G code number not supported.
0x021E	The coordinates instruction of robot system command is incorrect.	Syntax of [X, Y, Z] or [J1, J2, J3] command error. Correct the syntax format. Correct format: G01.1 X_Y_Z_A_B_C_ P0000 H0 R0 Q0 When Q is 0,1 or 2, use XYZABC. G01.1 J1_ J2_ J3_ P0000 H0 R0 Q3 When Q is 3, use J1, J2, …, J6.
0x021F	The tool number of robot system command is incorrect.	Syntax of H command error of H. Correct the syntax format. Correct format: G01.1 X_Y_Z_A_B_C_ P0000 H0 R0 Q0
0x0220	The workpiece number of robot system command is incorrect.	Syntax of R command error. Correct the syntax format. Correct format: G01.1 X_Y_Z_A_B_C_ P0000 H0 R0 Q0

Alarm code (Hex)	Name	Cause and correction
0x0221	The coordinates setting of robot system command is incorrect.	Syntax of Q command error. Correct the syntax format. Correct format: G01.1 X_Y_Z_A_B_C_ P0000 H0 R0 Q0 Q: setting range from 0 to 3.
0x0222	Command syntax error. No value specified in the instruction.	Syntax error. Correct the syntax format. Error example: G01 X Y100. ➔ No value specified after X.
0x0223	Command syntax error. Coordinate command value exceeds the range.	Syntax error. Correct the syntax format. Correct format: G01 X987654.321 G01 X98765432.1 The maximum number of digits is 9.
0x0224	Command syntax error. GOTO statement error.	Syntax error. Correct the syntax format. Error example: GOTO 2 N2 can't be found in the NC program.
0x0225	Command syntax error. END statement error.	Syntax error. Correct the syntax format. Error example: END No value specified after END.
0x0226	Command syntax error. DO statement error.	Syntax error. Correct the syntax format. Error example: WHILE DO No value specified after DO.
0x0229	Operational error. The specified file line number can't be found.	Error example: M98 Q10 Specifies line 10, but the NC program only contains 9 lines of content.
0x022A	Operational error. The specified N line number can't be found.	Error example: M98 H10 Specifies line 10, but there is no N10 line number instruction in the NC program.
0x022B	Operational error. The specified DO loop instruction can't be found.	Syntax error. Correct the syntax format. The END command doesn't have a corresponding DO.
0x022C	Operational error. The specified END can't be found.	The specified number of DO layers doesn't have corresponding END layer number. For example: While using WHILE DO 2, it should be ended with END 2.
0x022D	Operational error. Subprogram calling layers exceeds the maximum limit.	The maximum layer of subprogram call or macro stack is 8.
0x022E	Operational error. Missing P command in M98 subprogram calling function.	Missing P_ definition for the subprogram in M98 subprogram calling function.
0x022F	Operational error. Missing B command after G122.	B_ is not used with G122 command.
0x0230	Operational error. G122 B command misused.	B_ misused in G122 command.
0x0231	Operational error. The axis of drilling cycle is incorrect.	The axis used in the drilling cycle command doesn't exist.
0x0232	Operational error. Missing P_ or Q_ in the drilling cycle command.	The drilling cycle command doesn't apply P or Q.

A

Alarm code (Hex)	Name	Cause and correction
0x0233	Operational error. The drilling cycle command can't find the specified Q label.	The Q label defined by the drilling cycle command can't be found in the NC program.
0x0234	Operational error. Unable to find the specified N line number or file line number during breakpoint search.	While using breakpoint search, there is no corresponding N line number or file line number in the NC program.
0x0235	The # variable number exceeds the usage range.	Using an undefined # variable number or when the odd addresses are used between the # variable range #25128 ~ #25255 and #25384 ~ #25511 while using the floating-point MLC variable type with **[N1.010 Bit7=1]**, which occupies 32 bits of memory space.
0x0236	Error in the format of the extended workpiece coordinates.	The extended value for workpiece coordinates must be an integer. G54 P_ ➔ The parameter of P command must be an integer.
0x0237	The extended workpiece coordinates exceed the range.	The extended workpiece coordinates exceed the maximum limit. The range is G54 P1 to G54 P256.
0x0238	IF command error.	Correct the syntax of IF.
0x0239	THEN command error.	Correct the syntax of IF and THEN.
0x023A	ELSEIF command error.	Correct the syntax of IF and ELSEIF.
0x023B	ELSE command error.	Correct the syntax of IF and ELSE.
0x023C	ENDIF command error.	Correct the syntax of IF and ENDIF.
0x023D	Cannot find the specified ENDIF.	The NC program has IF statement but doesn't have the corresponding ENDIF.
0x023E	The layers used of IF command exceed the maximum limit.	The maximum layers used of IF command is 5.
0x0241	The P command is not marked.	The P command is not marked with the corresponding number.
0x0242	The value of P command exceeds the maximum value.	The limit value of P command is 1 to 256.
0x0243	The P command value is incorrect.	Syntax error. Correct the syntax format.
0x0244	The setting value of the # variable exceeds the limit range.	The setting value of the special # variable exceeds the allowed range. Error example: #20020 = 32768 (The value range of #20020 is 0 to 32767).
0x0245	The command P or Q is not used in the nested loop.	Syntax error. Correct the syntax format.
0x0246	The cutting amount in the roughing cycle is invalid.	Syntax error. Correct the syntax format.
0x0247	The nested loop must be followed by 2 consecutive blocks of instructions.	Syntax error. Correct the syntax format.
0x0248	The number of M codes used in a single block instruction exceeds the limit.	The maximum number of M codes allowed in a single block instruction is 4.
0x0249	M code usage error.	Syntax error. Correct the syntax format. Error example: M98 M1 M3
0x024B	Call instruction usage error.	Error example: M98 G65 One block can only have one call command.
0x024D	Multi Z axis application with tool compensation cannot specify the movement amount of the slave axis.	Syntax error. Correct the syntax format.

Alarm code (Hex)	Name	Cause and correction
0x024E	Usage error of the HSI value of multi-Z axis.	The trigger contact of G31 contains undefined contacts.
0x024F	Usage error of V_ or XYZABCUVW_ in the G31 skip function of multi-Z axis.	G31 command for the slave axis was used in the NC program.
0x0250	Usage error of the input function of the multi-Z axis in the G10 programmable parameter function.	Syntax error of G10 L50.
0x0251	Error parameters of the call subprogram function. Only P_, H(Q)_, L_, can be used.	Error example: M98 K5 K5 is not supported by M98.
0x0252	Usage error of read and write servo parameters of G10 programmable parameters function.	Syntax error of G10 L40 / G10 L41.
0x0253	Data error of the intermedia point for returning to the reference point.	Syntax error of G28.
0x0254	Attempt to assign a value to a ready-only # variable.	Error example: #20005 = 10 #20005 is a constant value π (3.1415) and cannot be written.
0x0255	The number of interpolation axes in a single block exceeds the limit.	The maximum number of interpolation axes in a single section is 16.
0x0256	Usage error of the arithmetic command.	Syntax error of the arithmetic command. Error example: #100=BIT[1] The BIT command requires 2 input values for calculation, but only one set of value is provided.
0x0257	Input numerical error in arithmetic command.	Syntax numerical error of the arithmetic command. Error example: B=ASIN[A] The range of A value is not between -1 to 1.
0x0258	Decimal point is not allowed.	The value of the command does not allow decimal points. Error example: G04 P1.2 The parameter of P does not support decimal values.
0x0259	The radius of the arc command is incorrect.	The distance between the center and the end point calculated by the arc command exceeds the arc radius tolerance of **N1.040**.
0x025A	The center of the arc command is incorrect.	The distance between the center and the end point calculated by the arc command exceeds the arc radius tolerance of **N1.040**.
0x025B	3-dimension arc command error.	The distance between the center and the end point calculated by the 3-dimension arc command exceeds the arc radius tolerance of **N1.040**.
0x025C	Blending function data registration error	G10 L60 blending function data registration error. Please check the value of the P command.
0x025D	Working plane setting error.	While enabling synchronize axis function, the plane selections such as G17, G18 and G19 have the same axis, or the plane selection is incorrect.

A

Alarm code (Hex)	Name	Cause and correction
0x025E	System obtained NULL pointers.	RBTask obtained NULL pointers. Controller malfunction, please contact Delta or supplier for assistance.
0x025F	Not supported instruction for the scaling function.	When using the G68 rotation command, it is not allowed to use 3-dimension arc command, G28/G29/G30, and G54~G59 commands simultaneously. If you need to use them, you must first cancel the G68 function with G69.
0x0260	Insufficient information for circle command by three points method.	The input information used for circle command by three points is insufficient.
0x0261	Subprogram calling function missing H_ statement.	Syntax error. Correct the syntax format.
0x0262	Switching arm gesture is prohibited during arm moving.	Switching hand orientation is prohibited during motion.
0x0263	File not found.	Cannot find the specified NC program file.
0x0264	The P_Q_ command does not exist in looping command.	Make sure the NC program file includes the N line number or file line number for specified by P_Q_.
0x0265	G53 or G28/ G30 commands error	G53 and G28/G30 commands cannot be used in the same block.

A.2.2 NC Axis Alarms

Alarm code (Hex)	Name	Cause and correction
0x0401	Emergency stop signal is triggered.	Exclude the EMG signal.
0x0402	Hardware limit signal is triggered.	Exclude the hardware limit signal.
0x0601	Axes reference information error	Communication is disconnected after established. Reset after reconnecting the physical cable. When the alarm occurs, the origin state of the axis will be canceled if it's an incremental type of axis.
0x0602	Overspeed protection.	System overspeed protection. The axis will servo off when the alarm occurs. It will be automatically servo on again after reset command and clearing the alarm.
0x0A01	The 1st segment of software limit is triggered.	The axis has reached the software limit position of 1st segment. The machine must move away from the limit position before system reset.
0x0A02	The status of 1st segment of software limit needs to be cleared.	After moving away from the 1st segment of software limit, this alarm will be displayed. Please reset the machine.
0x0A03	The 2nd segment of software limit is triggered.	The axis has reached the software limit position of 2nd segment. The machine must move away from the limit position before system reset.
0x0A04	The status of 2nd segment of software limit needs to be cleared.	After moving away from the 2nd segment of software limit, this alarm will be displayed. Please reset the machine.
0x0A05	Hardware limit triggered.	The axis has reached the hardware limit position. The machine must move away from the limit position before system reset.
0x0A06	The status of hardware limit needs to be cleared.	After moving away from the hardware limit, this alarm will be displayed. Please reset the machine.
0x0A07	Servo is not ready.	The axis is not servo ON.
0x0A08	Origin status of the axis is missing.	Please execute the homing operation.
0x0C01	Spindle positioning failed.	Verify the spindle positioning process or check if the spindle positioning time setting is incorrect. **N0.1025**, Positioning check time for 1st spindle. **N0.1075**, Positioning check time for 2nd spindle. **N0.1125**, Positioning check time for 3rd spindle. … **N0.1375**, Positioning check time for 8th spindle.
0x0C02	Spindle is not rotating before cutting.	**[N1.011 bit8 the spindle enables checks before cutting]** is set to 1, and the spindle is not rotating when executing the motion command.
0x0C03	The spindle speed setting exceeds the maximum speed.	**[N0.1000 bit7 the spindle maximum speed command check]** is set to 1, and the S value in the NC instruction exceeds the maximum spindle speed setting in **N0.1008**.
0x0C04	Incorrect switching of the spindle and C-axis.	Make sure all channel parameters are set, whether the C-axis and the spindle are both configured as active axes.
0x0C05	The spindle C-axis is not set as rotational axis.	Check the C axis setting of **N2.001 bits 2 to 4** for the rotational axis feed mode.
0x0C06	The spindle is not enabled.	Please check if the spindle is enabled.

A

0x1601	The hardware limit is triggered during execution of home mode 4. The search direction does not match.	The hardware limit is triggered during homing. Please check the following: 1. Make sure the machine is out of any limit positions before executing the homing procedure. 2. Verify the wiring signals for the hardware limits are functioning correctly. 3. Chek if the homing search direction is correct.
0x1602	The hardware limit alarm not cleared of the home mode 4.	This alarm will be triggered after clearing 0x1601 hardware limit alarm.
0x1603	Oscillation function is not allowed in the home mode.	The M2x264 oscillation function cannot be enabled in the homing mode.
0x1604	The home mode is not supported.	Please confirm the parameter settings of **N2.50** homing mode.
0x1605	The axis is set as non-rotary axis in home mode 6.	Please confirm the parameter settings of **N2.01** axis configuration.
0xF002	The alarm codes of Delta servo.	The alarm of 0xF002 needs to be interpreted in conjunction with the alarm codes of the Delta servo. For example, if the Delta servo displays an alarm code 0x09 (following error), the controller will display 0xF002 and indicate the servo's alarm code as 0x09.

A.2.3 NC Motion Interpolation Alarms

Alarm code (Hex)	Name	Cause and correction
0x1001	Feed rate is not specified.	The NC program needs to specify F or set default feed rate value.
0x1201	Block command error.	Command block error.
0x1202	The repeated points of tool compensation have exceeded the maximum limit.	The repeated points exceed the maximum limit.
0x1203	Plane changing after enabling tool compensation.	Changing the plane is prohibited after enabling tool compensation.
0x1204	Unable to calculate the tool compensation path at arc-to-linear command.	Unable to calculate the tool compensation path.
0x1205	Tool compensation path interference occurred at arc-to-linear.	Tool compensation result has path interference. For example: The arc entry is smaller than the tool diameter.
0x1206	Changing tool diameter during arc command performing.	Tool diameter cannot be changed during arc command while tool compensation is enabled.
0x1207	Unable to calculate the tool compensation path at arc-to-arc command.	Unable to calculate the tool compensation path.
0x1208	Tool compensation path interference occurred between linear-to-arc.	Tool compensation result has path interference. For example: The arc entry is smaller than the tool diameter.
0x1209	Unable to calculate the tool compensation path at linear-to-arc command.	Unable to calculate the tool compensation pat.
0x120A	The arc radius is less than 0 after tool compensation.	Tool compensation result has path interference.
0x120B	Tool compensation path interference occurred at two linear commands.	Tool compensation result has path interference.
0x120C	Calculation error at the arm path interpolation.	The endpoint position is outside the working range of the robot arm.
0x120D	The P value of G05 command does not correspond to HMI recipe.	The corresponding recipe values must be filled in within the HMI interface.
0x120E	Path interpolation distance exceeds the maximum travel distance.	Motion coordinate point exceeds the maximum distance. Error example: X[tan[90]] ➔ tan[90] is a infinity value Please correct the coordinate position of the motion command.

A.2.4 NC Kernel Alarms

A

Alarm code (Hex)	Name	Cause and correction
0x1401	System kernel construction failed.	Controller malfunction, please contact Delta or supplier for assistance.
0x1402	System kernel RT construction failed.	Controller malfunction, please contact Delta or supplier for assistance.
0x1403	System kernel RT memory error.	Controller malfunction, please contact Delta or supplier for assistance.
0x1404	System RT deployment configuration error.	Controller malfunction, please contact Delta or supplier for assistance.
0x1405	Path UTY construction failed.	Controller malfunction, please contact Delta or supplier for assistance.
0x1406	Path NRT construction failed.	Controller malfunction, please contact Delta or supplier for assistance.
0x1407	Path RT construction failed.	Controller malfunction, please contact Delta or supplier for assistance.
0x1408	Path UTY memory error.	Controller malfunction, please contact Delta or supplier for assistance.
0x1409	Path NRT memory error.	Controller malfunction, please contact Delta or supplier for assistance.
0x140A	Path RT memory error.	Controller malfunction, please contact Delta or supplier for assistance.
0x140B	Retentive # variable error.	Controller malfunction, please contact Delta or supplier for assistance.
0x140C	Path UTY deployment parameter error.	Parameter error.
0x140D	Path UTY deployment configuration error.	Parameter error.
0x140E	Path NRT association error.	Controller malfunction, please contact Delta or supplier for assistance.
0x140F	Path NRT deployment parameter error.	Parameter error.
0x1410	Path NRT deployment configuration error.	Parameter error.
0x1411	Path RT association error.	Controller malfunction, please contact Delta or supplier for assistance.
0x1412	Path RT deployment parameter error.	Parameter error.
0x1413	Path RT deployment configuration error.	Parameter error.
0x1414	Path NRT registration # variable error.	Controller malfunction, please contact Delta or supplier for assistance.
0x1415	Virtual axis type error.	The system axis parameters have both physical and virtual axis setting on the same axis.
0x1416	The slave address does not match.	The slave address is different from the system address setting. Please confirm the slave address or reset.
0x1417	The slave address is duplicated.	Duplicate slave addresses are being used. Please check the address setting of the slave devices and eliminate the duplicate addresses.
0x1418	The number of channels in the multi-channel is incorrect.	The NC multi-channel must be used sequentially and cannot be used with only channel one and channel three.
0x1419	NC axis number setting error.	Controller malfunction, please contact Delta or supplier for assistance.
0x141A	Spindle number setting error.	Controller malfunction, please contact Delta or supplier for assistance.
0x141B	Coding of the tool magazine setting error.	Controller malfunction, please contact Delta or supplier for assistance.

0x141C	Setting error of the axis type.	The axis type setting, such as NC axis or PLC axis, is incorrect. Please contact Delta or supplier for assistance.
0x141D	The number of Cartesian axes and joint axes in the path dose not match.	Controller malfunction, please contact Delta or supplier for assistance.
0x141E	System PDO mapping mismatch.	Controller malfunction, please contact Delta or supplier for assistance. The system DAT configuration does not match the connected products.
0x141F	The system interrupt execution time is not supported.	The servo does not support the current interpolation time.
0x1420	EtherCAT communication establish failed.	Check if the communication cables are properly connected to the slave devices.
0x1426	The system remote module (EIO) failed.	Verify if there are any conflicts with the settings in the controller EIO interface and confirm the changes to save the modified parameter values.
0x1427	Failed to trigger the EtherCAT communication OP mode.	Check if the fieldbus is properly connected.
0x1428	Failed to switch the EtherCAT communication to OP mode.	Check if the fieldbus is properly connected.
0x1429	EtherCAT communication initialization error.	Please restart the system.
0x142A	The number of axes defined in the channel exceeds the limit.	Please check the parameter definition of the channel axis.
0x142B	EtherCAT slave devices not found.	Check if the cables are properly connected.
0x142C	Conflicting setting between multi-Z and synchronization function.	Check for any conflicts between the multi-Z parameters and the synchronization control parameter **N2.015**.
0x142D	Setting error of the multi-Z machine parameters.	Check if the setting of multi-Z parameters is correct.
0x142E	The system remote module (EIO) connection device type mismatch with the settings.	Check if the EIO module type settings match the devices on the fieldbus.
0x142F	The corresponding port number for the remote module (EIO) connection was not found.	Check if the port number setting for the EIO module is correct.

Special Memory

B

This chapter introduces CNC system definition device list. It supplies users to check MLC special # variable action correct or not in CNC system.

B.1 Definition of MLC special M relay and special D register

The motion logic control (MLC) and the NC are two independent systems. The MLC system performs button triggering control, MLC axis movements, and other logic controls, while the NC system manages functions related to the system and servo axis. The MLC's special M relays and D registers serve as the I/O interface between these two systems for data exchange and signal transmission. The output mentioned in this chapter refers to the signals sent to the NC system from the MLC special M relays and D registers. The input refers to the signals sent to the MLC special M relays and D registers from the NC system.

The M letter prefixed codes are in bit format, being 0 (OFF) and 1 (ON). The D prefixed codes are in word format, referring to numerical values such as 1000. The MLC special M and D codes are all expressed in the form of M- and D- prefixes followed by five digits.

Data exchanges between the two systems are categorized into four groups.
1: MLC bit output from MLC to NC (special M, bit output)
2: MLC bit input from NC to MLC (special M, bit input)
3: MLC word output from MLC to NC (special D, word output)
4: MLC word input from NC to MLC (special D, word input)

There are each 65,535 address numbers for the M and D registers in the NC5 series controller. The range between M20000 to M24999, M30000 to M34999, D20000 to D24999 and D30000 to D34999 are the special M and D registers in the system. Therefore, all of these special M and D register will separated in different groups according to the NC5 series controller support multi-channel control, as shown in the table below.

Channel Index	System Special M and D MLC to System (NC)		System Special M and D System (NC) to MLC	
General	M20000 ~ M20999	D20000 ~ D20999	M30000 ~ M30999	D30000 ~ D30999
Channel 1	M21000 ~ M21999	D21000 ~ D21999	M31000 ~ M31999	D31000 ~ D31999
Channel 2	M22000 ~ M22999	D22000 ~ D22999	M32000 ~ M32999	D32000 ~ D32999
Channel 3	M23000 ~ M23999	D23000 ~ D23999	M33000 ~ M33999	D33000 ~ D33999
Channel 4	M24000 ~ M24999	D24000 ~ D24999	M34000 ~ M34999	D34000 ~ D34999

B

Channel Index	For HMI Special M and D MLC to System (HMI)		For HMI Special M and D System (HMI) to MLC	
General	M49000 ~ M49099	D49000 ~ D49099	M59000 ~ M59099	D59000 ~ D59099
Channel 1	M49100 ~ M49199	D49100 ~ D49199	M59100 ~ M59199	D59100 ~ D59199
Channel 2	M49200 ~ M49299	D49200 ~ D49299	M59200 ~ M59299	D59200 ~ D59299
Channel 3	M49300 ~ M49399	D49300 ~ D49399	M59300 ~ M59399	D59300 ~ D59399
Channel 4	M49400 ~ M49499	D49400 ~ D49499	M59400 ~ M59499	D59400 ~ D59499

For the special M and D register, when the tens of thousands digit is 2, it means the MLC is sending a value to the NC system. If the digit is 3, it means the NC system is updating its status for the MLC to read and check. If the digit is 4, it means the MLC is sending a value to the HMI system. If the digit is 5, it means the HMI system is updating its status for the MLC to read and check. When the tens of thousands digit is 2 or 3 and the thousands digit is 0, it means this register is for system general purpose use. If the digit is 1 to 4, it represents the specific NC channel respectively. The remaining three digits from 000 to 999 are for the functional index. The chapters below will give detailed function descriptions for each available special M and D.

B.2 List of special M and special D

B.2.1 List of special M (MLC to system)

B.2.1.1 M21000 to M29999

Function Name	Special M	Description	Device
Spindle1 DO Control	M20018	When this function is ON, the system will output the digital signal on the Pin 9 of Spindle 1 connector.	R/W
Spindle2 DO Control	M20019	When this function is ON, the system will output the digital signal on the Pin 9 of Spindle 2 connector.	R/W
Enable 3 MPG Control	M20024	Enable 3 sets of MPG function control. The controller can process three sets of MPG pulse signals and control the three axes respectively. When this special M is enabled, the 2nd and 3rd MPG will also be enabled at the same time.	R/W
Cycle Start	M2x000	Informs the NC system to execute Cycle Start and execute the NC program.	R/W
Feed Hold	M2x001	Pause the NC process in the controller system. After pausing, the system can resume the procedure with “Cycle Start” or stop all actions with “Reset”.	R/W
NC Reset	M2x004	Tells the NC system to execute the Reset procedure; all actions will be stopped.	R/W
Enable Dry Run	M2x005	Enable dry run mode. The system will execute NC programs, including regular feed and rapid command, at dry run speed. The dry run speed will refer to parameter **[N1.013 Bit18 Dry run speed mode]** and **[N1.66 Dry run feed rate]** for the speed reference. Parameter / Setting / Actual Speed N1.013 Bit18 / 0 / N1.66 Setting N1.013 Bit18 / 1 / N1.66 x D2x002	R/W
MPG Simulation	M2x006	During program execution, users can use the MPG to control the speed of movement trajectories.	R/W
Disable Hardware Limit	M2x007	The limit signal of each axis is ignored when this function is enabled.	R/W
Single Block	M2x008	In AUTO mode, the program stops after one block is executed.	R/W
Optional Stop	M2x009	Enable the optional stop key. When the program executes M01, the controller immediately stops.	R/W
Single Block Skip (‘/’)	M2x010	The program skips the block containing the symbol '/' when this function is enabled.	R/W
M, S, and T Codes Lock	M2x011	The program skips any block containing M, S, T code when this function is enabled.	R/W
Servo ON/ OFF	M2x012	Servo ON or OFF for all of connected servo drives in the corresponding channel.	R/W
Enable Emergency Stop	M2x013	When this function is ON, the NC will trigger an emergency stop.	R/W
Enable Synchronization Adjustment	M2x014	Manually trigger synchronization adjustment.	R/W
M, S, and T Codes Finished	M2x016	Triggering this signal informs the NC system that the procedures for M, S and T codes are complete.	R/W
M96 Program Interruption	M2x019	After the NC executes M96, if this function is triggered, the NC system interrupts the main program and jumps to the subprogram to execute it.	R/W
M, S, and T Codes Call Macro Lock	M2x021	When this function is ON, the NC will not execute macro program including M code process regarding M, S, and T codes in the NC program.	R/W
Macro Call Activation	M2x025	Activates macro call. (Only works with the correct macro-ID in AUTO mode)	R/W

B

<table>
<tr><th>Function Name</th><th>Special M</th><th>Description</th><th>Device</th></tr>
<tr><td>M99 Call Stop</td><td>M2x026</td><td>When this function is ON, the NC system will slow down to stop and then stop all process when executing M99.</td><td>R/W</td></tr>
<tr><td>1st Macro Call Preparation</td><td>M2x032</td><td rowspan="16">Initializes macro call.
(A successful call will only occur when the correct macro number [D2x064~D2x079] and the corresponding macro program both exist)</td><td>R/W</td></tr>
<tr><td>2nd Macro Call Preparation</td><td>M2x033</td><td>R/W</td></tr>
<tr><td>3rd Macro Call Preparation</td><td>M2x034</td><td>R/W</td></tr>
<tr><td>4th Macro Call Preparation</td><td>M2x035</td><td>R/W</td></tr>
<tr><td>5th Macro Call Preparation</td><td>M2x036</td><td>R/W</td></tr>
<tr><td>6th Macro Call Preparation</td><td>M2x037</td><td>R/W</td></tr>
<tr><td>7th Macro Call Preparation.</td><td>M2x038</td><td>R/W</td></tr>
<tr><td>8th Macro Call Preparation</td><td>M2x039</td><td>R/W</td></tr>
<tr><td>9th Macro Call Preparation</td><td>M2x040</td><td>R/W</td></tr>
<tr><td>10th Macro Call Preparation</td><td>M2x041</td><td>R/W</td></tr>
<tr><td>11th Macro Call Preparation</td><td>M2x042</td><td>R/W</td></tr>
<tr><td>12th Macro Call Preparation</td><td>M2x043</td><td>R/W</td></tr>
<tr><td>13th Macro Call Preparation</td><td>M2x044</td><td>R/W</td></tr>
<tr><td>14th Macro Call Preparation</td><td>M2x045</td><td>R/W</td></tr>
<tr><td>15th Macro Call Preparation</td><td>M2x046</td><td>R/W</td></tr>
<tr><td>16th Macro Call Preparation</td><td>M2x047</td><td>R/W</td></tr>
<tr><td rowspan="4">Robot Coordinate System</td><td>M2x048</td><td rowspan="4">The robot coordinate system can be set as shown in the table below.
The CNC controller needs to be in manual mode. The D2x016 defines the coordinate system, eg.: 1 = G54, 2 = G55, …, 5 = G58 and 6 = G59.
<table>
<tr><th></th><th>PCS (Piece)</th><th>TCS (Tool)</th><th>JCS (Joint)</th></tr>
<tr><td>M2x048</td><td>1</td><td>1</td><td>0</td></tr>
<tr><td>M2x049</td><td>1</td><td>1</td><td>0</td></tr>
<tr><td>M2x050</td><td>1</td><td>0</td><td>0</td></tr>
<tr><td>M2x051</td><td>1</td><td>1</td><td>0</td></tr>
<tr><td>D2x016</td><td>1 ~ 6</td><td>1 ~ 6</td><td>1 ~ 6</td></tr>
</table></td><td rowspan="4">R/W</td></tr>
<tr><td>M2x049</td></tr>
<tr><td>M2x050</td></tr>
<tr><td>M2x051</td></tr>
<tr><td>Tool Magazine 1 Move Forward</td><td>M2x064</td><td>Moves tool magazine 1 forward. When this special M relay is triggered, the tool pot deviation [D3x039] is decreased by 1, and the standby tool pot number [D3x038] is increased by 1.</td><td>R/W</td></tr>
<tr><td>Tool Magazine 1 Move Backward</td><td>M2x065</td><td>Moves tool magazine 1 backward. When this special M relay is triggered, the tool pot deviation [D3x039] is increased by 1, and the standby tool pot number [D3x038] is decreased by 1.</td><td>R/W</td></tr>
<tr><td>Tool 1 Exchange</td><td>M2x066</td><td>Exchanges tool data in tool magazine 1.</td><td>R/W</td></tr>
<tr><td>Tool Magazine 1 Reset</td><td>M2x067</td><td>When this special M relay is triggered, the tool data in tool magazine 1 is reset.</td><td>R/W</td></tr>
</table>

Function Name	Special M	Description	Device
Tool Magazine 2 Move Forward	M2x072	Moves tool magazine 1 forward. When this special M relay is triggered, the tool pot deviation **[D3x045]** is decreased by 1, and the standby tool pot number **[D3x044]** is increased by 1.	R/W
Tool Magazine 2 Move Backward	M2x073	Moves tool magazine 1 backward. When this special M relay is triggered, the tool pot deviation **[D3x045]** is increased by 1, and the standby tool pot number **[D3x044]** is decreased by 1.	R/W
Tool 2 Exchange	M2x074	Exchanges tool data in tool magazine 2.	R/W
Tool Magazine 2 Reset	M2x075	When this special M relay is triggered, the tool data in tool magazine 2 is reset.	R/W
Canceling Tapping Interrupt Status	M2x080	When the NC system is in the tapping interrupt status, users can trigger this M relay to cancel the interrupt status.	R/W
MLC to NC Variable 1	M2x128	The system will move the binary status of this special M to NC variable #25000 as 0 or 1.	R/W
MLC to NC Variable 2	M2x129	The system will move the binary status of this special M to NC variable #25001 as 0 or 1.	R/W
MLC to NC Variable 3	M2x130	The system will move the binary status of this special M to NC variable #25002 as 0 or 1.	R/W
MLC to NC Variable 4	M2x131	The system will move the binary status of this special M to NC variable #25003 as 0 or 1.	R/W
MLC to NC Variable 5	M2x132	The system will move the binary status of this special M to NC variable #25004 as 0 or 1.	R/W
MLC to NC Variable 6	M2x133	The system will move the binary status of this special M to NC variable #25005 as 0 or 1.	R/W
MLC to NC Variable 7	M2x134	The system will move the binary status of this special M to NC variable #25006 as 0 or 1.	R/W
MLC to NC Variable 8	M2x135	The system will move the binary status of this special M to NC variable #25007 as 0 or 1.	R/W
MLC to NC Variable 9	M2x136	The system will move the binary status of this special M to NC variable #25008 as 0 or 1.	R/W
MLC to NC Variable 10	M2x137	The system will move the binary status of this special M to NC variable #25009 as 0 or 1.	R/W
MLC to NC Variable 11	M2x138	The system will move the binary status of this special M to NC variable #25010 as 0 or 1.	R/W
MLC to NC Variable 12	M2x139	The system will move the binary status of this special M to NC variable #25011 as 0 or 1.	R/W
MLC to NC Variable 13	M2x140	The system will move the binary status of this special M to NC variable #25012 as 0 or 1.	R/W
MLC to NC Variable 14	M2x141	The system will move the binary status of this special M to NC variable #25013 as 0 or 1.	R/W
MLC to NC Variable 15	M2x142	The system will move the binary status of this special M to NC variable #25014 as 0 or 1.	R/W
MLC to NC Variable 16	M2x143	The system will move the binary status of this special M to NC variable #25015 as 0 or 1.	R/W
MLC to NC Variable 17	M2x144	The system will move the binary status of this special M to NC variable #25016 as 0 or 1.	R/W
MLC to NC Variable 18	M2x145	The system will move the binary status of this special M to NC variable #25017 as 0 or 1.	R/W
MLC to NC Variable 19	M2x146	The system will move the binary status of this special M to NC variable #25018 as 0 or 1.	R/W
MLC to NC Variable 20	M2x147	The system will move the binary status of this special M to NC variable #25019 as 0 or 1.	R/W
MLC to NC Variable 21	M2x148	The system will move the binary status of this special M to NC variable #25020 as 0 or 1.	R/W

B

Function Name	Special M	Description	Device
MLC to NC Variable 22	M2x149	The system will move the binary status of this special M to NC variable #25021 as 0 or 1.	R/W
MLC to NC Variable 23	M2x150	The system will move the binary status of this special M to NC variable #25022 as 0 or 1.	R/W
MLC to NC Variable 24	M2x151	The system will move the binary status of this special M to NC variable #25023 as 0 or 1.	R/W
MLC to NC Variable 25	M2x152	The system will move the binary status of this special M to NC variable #25024 as 0 or 1.	R/W
MLC to NC Variable 26	M2x153	The system will move the binary status of this special M to NC variable #25025 as 0 or 1.	R/W
MLC to NC Variable 27	M2x154	The system will move the binary status of this special M to NC variable #25026 as 0 or 1.	R/W
MLC to NC Variable 28	M2x155	The system will move the binary status of this special M to NC variable #25027 as 0 or 1.	R/W
MLC to NC Variable 29	M2x156	The system will move the binary status of this special M to NC variable #25028 as 0 or 1.	R/W
MLC to NC Variable 30	M2x157	The system will move the binary status of this special M to NC variable #25029 as 0 or 1.	R/W
MLC to NC Variable 31	M2x158	The system will move the binary status of this special M to NC variable #25030 as 0 or 1.	R/W
MLC to NC Variable 32	M2x159	The system will move the binary status of this special M to NC variable #25031 as 0 or 1.	R/W
MLC to NC Variable 33	M2x160	The system will move the binary status of this special M to NC variable #25032 as 0 or 1.	R/W
MLC to NC Variable 34	M2x161	The system will move the binary status of this special M to NC variable #25033 as 0 or 1.	R/W
MLC to NC Variable 35	M2x162	The system will move the binary status of this special M to NC variable #25034 as 0 or 1.	R/W
MLC to NC Variable 36	M2x163	The system will move the binary status of this special M to NC variable #25035 as 0 or 1.	R/W
MLC to NC Variable 37	M2x164	The system will move the binary status of this special M to NC variable #25036 as 0 or 1.	R/W
MLC to NC Variable 38	M2x165	The system will move the binary status of this special M to NC variable #25037 as 0 or 1.	R/W
MLC to NC Variable 39	M2x166	The system will move the binary status of this special M to NC variable #25038 as 0 or 1.	R/W
MLC to NC Variable 40	M2x167	The system will move the binary status of this special M to NC variable #25039 as 0 or 1.	R/W
MLC to NC Variable 41	M2x168	The system will move the binary status of this special M to NC variable #25040 as 0 or 1.	R/W
MLC to NC Variable 42	M2x169	The system will move the binary status of this special M to NC variable #25041 as 0 or 1.	R/W
MLC to NC Variable 43	M2x170	The system will move the binary status of this special M to NC variable #25042 as 0 or 1.	R/W
MLC to NC Variable 44	M2x171	The system will move the binary status of this special M to NC variable #25043 as 0 or 1.	R/W
MLC to NC Variable 45	M2x172	The system will move the binary status of this special M to NC variable #25044 as 0 or 1.	R/W
MLC to NC Variable 46	M2x173	The system will move the binary status of this special M to NC variable #25045 as 0 or 1.	R/W
MLC to NC Variable 47	M2x174	The system will move the binary status of this special M to NC variable #25046 as 0 or 1.	R/W
MLC to NC Variable 48	M2x175	The system will move the binary status of this special M to NC variable #25047 as 0 or 1.	R/W
MLC to NC Variable 49	M2x176	The system will move the binary status of this special M to NC variable #25048 as 0 or 1.	R/W

Function Name	Special M	Description	Device
MLC to NC Variable 50	M2x177	The system will move the binary status of this special M to NC variable #25049 as 0 or 1.	R/W
MLC to NC Variable 51	M2x178	The system will move the binary status of this special M to NC variable #25050 as 0 or 1.	R/W
MLC to NC Variable 52	M2x179	The system will move the binary status of this special M to NC variable #25051 as 0 or 1.	R/W
MLC to NC Variable 53	M2x180	The system will move the binary status of this special M to NC variable #25052 as 0 or 1.	R/W
MLC to NC Variable 54	M2x181	The system will move the binary status of this special M to NC variable #25053 as 0 or 1.	R/W
MLC to NC Variable 55	M2x182	The system will move the binary status of this special M to NC variable #25054 as 0 or 1.	R/W
MLC to NC Variable 56	M2x183	The system will move the binary status of this special M to NC variable #25055 as 0 or 1.	R/W
MLC to NC Variable 57	M2x184	The system will move the binary status of this special M to NC variable #25056 as 0 or 1.	R/W
MLC to NC Variable 58	M2x185	The system will move the binary status of this special M to NC variable #25057 as 0 or 1.	R/W
MLC to NC Variable 59	M2x186	The system will move the binary status of this special M to NC variable #25058 as 0 or 1.	R/W
MLC to NC Variable 60	M2x187	The system will move the binary status of this special M to NC variable #25059 as 0 or 1.	R/W
MLC to NC Variable 61	M2x188	The system will move the binary status of this special M to NC variable #25060 as 0 or 1.	R/W
MLC to NC Variable 62	M2x189	The system will move the binary status of this special M to NC variable #25061 as 0 or 1.	R/W
MLC to NC Variable 63	M2x190	The system will move the binary status of this special M to NC variable #25062 as 0 or 1.	R/W
MLC to NC Variable 64	M2x191	The system will move the binary status of this special M to NC variable #25063 as 0 or 1.	R/W
MLC to NC Variable 65	M2x192	The system will move the binary status of this special M to NC variable #25064 as 0 or 1.	R/W
MLC to NC Variable 66	M2x193	The system will move the binary status of this special M to NC variable #25065 as 0 or 1.	R/W
MLC to NC Variable 67	M2x194	The system will move the binary status of this special M to NC variable #25066 as 0 or 1.	R/W
MLC to NC Variable 68	M2x195	The system will move the binary status of this special M to NC variable #25067 as 0 or 1.	R/W
MLC to NC Variable 69	M2x196	The system will move the binary status of this special M to NC variable #25068 as 0 or 1.	R/W
MLC to NC Variable 70	M2x197	The system will move the binary status of this special M to NC variable #25069 as 0 or 1.	R/W
MLC to NC Variable 71	M2x198	The system will move the binary status of this special M to NC variable #25070 as 0 or 1.	R/W
MLC to NC Variable 72	M2x199	The system will move the binary status of this special M to NC variable #25071 as 0 or 1.	R/W
MLC to NC Variable 73	M2x200	The system will move the binary status of this special M to NC variable #25072 as 0 or 1.	R/W
MLC to NC Variable 74	M2x201	The system will move the binary status of this special M to NC variable #25073 as 0 or 1.	R/W
MLC to NC Variable 75	M2x202	The system will move the binary status of this special M to NC variable #25074 as 0 or 1.	R/W
MLC to NC Variable 76	M2x203	The system will move the binary status of this special M to NC variable #25075 as 0 or 1.	R/W
MLC to NC Variable 77	M2x204	The system will move the binary status of this special M to NC variable #25076 as 0 or 1.	R/W

B

Function Name	Special M	Description	Device
MLC to NC Variable 78	M2x205	The system will move the binary status of this special M to NC variable #25077 as 0 or 1.	R/W
MLC to NC Variable 79	M2x206	The system will move the binary status of this special M to NC variable #25078 as 0 or 1.	R/W
MLC to NC Variable 80	M2x207	The system will move the binary status of this special M to NC variable #25079 as 0 or 1.	R/W
MLC to NC Variable 81	M2x208	The system will move the binary status of this special M to NC variable #25080 as 0 or 1.	R/W
MLC to NC Variable 82	M2x209	The system will move the binary status of this special M to NC variable #25081 as 0 or 1.	R/W
MLC to NC Variable 83	M2x210	The system will move the binary status of this special M to NC variable #25082 as 0 or 1.	R/W
MLC to NC Variable 84	M2x211	The system will move the binary status of this special M to NC variable #25083 as 0 or 1.	R/W
MLC to NC Variable 85	M2x212	The system will move the binary status of this special M to NC variable #25084 as 0 or 1.	R/W
MLC to NC Variable 86	M2x213	The system will move the binary status of this special M to NC variable #25085 as 0 or 1.	R/W
MLC to NC Variable 87	M2x214	The system will move the binary status of this special M to NC variable #25086 as 0 or 1.	R/W
MLC to NC Variable 88	M2x215	The system will move the binary status of this special M to NC variable #25087 as 0 or 1.	R/W
MLC to NC Variable 89	M2x216	The system will move the binary status of this special M to NC variable #25088 as 0 or 1.	R/W
MLC to NC Variable 90	M2x217	The system will move the binary status of this special M to NC variable #25089 as 0 or 1.	R/W
MLC to NC Variable 91	M2x218	The system will move the binary status of this special M to NC variable #25090 as 0 or 1.	R/W
MLC to NC Variable 92	M2x219	The system will move the binary status of this special M to NC variable #25091 as 0 or 1.	R/W
MLC to NC Variable 93	M2x220	The system will move the binary status of this special M to NC variable #25092 as 0 or 1.	R/W
MLC to NC Variable 94	M2x221	The system will move the binary status of this special M to NC variable #25093 as 0 or 1.	R/W
MLC to NC Variable 95	M2x222	The system will move the binary status of this special M to NC variable #25094 as 0 or 1.	R/W
MLC to NC Variable 96	M2x223	The system will move the binary status of this special M to NC variable #25095 as 0 or 1.	R/W
MLC to NC Variable 97	M2x224	The system will move the binary status of this special M to NC variable #25096 as 0 or 1.	R/W
MLC to NC Variable 98	M2x225	The system will move the binary status of this special M to NC variable #25097 as 0 or 1.	R/W
MLC to NC Variable 99	M2x226	The system will move the binary status of this special M to NC variable #25098 as 0 or 1.	R/W
MLC to NC Variable 100	M2x227	The system will move the binary status of this special M to NC variable #25099 as 0 or 1.	R/W
MLC to NC Variable 101	M2x228	The system will move the binary status of this special M to NC variable #25100 as 0 or 1.	R/W
MLC to NC Variable 102	M2x229	The system will move the binary status of this special M to NC variable #25101 as 0 or 1.	R/W
MLC to NC Variable 103	M2x230	The system will move the binary status of this special M to NC variable #25102 as 0 or 1.	R/W
MLC to NC Variable 104	M2x231	The system will move the binary status of this special M to NC variable #25103 as 0 or 1.	R/W
MLC to NC Variable 105	M2x232	The system will move the binary status of this special M to NC variable #25104 as 0 or 1.	R/W

Function Name	Special M	Description	Device
MLC to NC Variable 106	M2x233	The system will move the binary status of this special M to NC variable #25105 as 0 or 1.	R/W
MLC to NC Variable 107	M2x234	The system will move the binary status of this special M to NC variable #25106 as 0 or 1.	R/W
MLC to NC Variable 108	M2x235	The system will move the binary status of this special M to NC variable #25107 as 0 or 1.	R/W
MLC to NC Variable 109	M2x236	The system will move the binary status of this special M to NC variable #25108 as 0 or 1.	R/W
MLC to NC Variable 110	M2x237	The system will move the binary status of this special M to NC variable #25109 as 0 or 1.	R/W
MLC to NC Variable 111	M2x238	The system will move the binary status of this special M to NC variable #25110 as 0 or 1.	R/W
MLC to NC Variable 112	M2x239	The system will move the binary status of this special M to NC variable #25111 as 0 or 1.	R/W
MLC to NC Variable 113	M2x240	The system will move the binary status of this special M to NC variable #25112 as 0 or 1.	R/W
MLC to NC Variable 114	M2x241	The system will move the binary status of this special M to NC variable #25113 as 0 or 1.	R/W
MLC to NC Variable 115	M2x242	The system will move the binary status of this special M to NC variable #25114 as 0 or 1.	R/W
MLC to NC Variable 116	M2x243	The system will move the binary status of this special M to NC variable #25115 as 0 or 1.	R/W
MLC to NC Variable 117	M2x244	The system will move the binary status of this special M to NC variable #25116 as 0 or 1.	R/W
MLC to NC Variable 118	M2x245	The system will move the binary status of this special M to NC variable #25117 as 0 or 1.	R/W
MLC to NC Variable 119	M2x246	The system will move the binary status of this special M to NC variable #25118 as 0 or 1.	R/W
MLC to NC Variable 120	M2x247	The system will move the binary status of this special M to NC variable #25119 as 0 or 1.	R/W
MLC to NC Variable 121	M2x248	The system will move the binary status of this special M to NC variable #25120 as 0 or 1.	R/W
MLC to NC Variable 122	M2x249	The system will move the binary status of this special M to NC variable #25121 as 0 or 1.	R/W
MLC to NC Variable 123	M2x250	The system will move the binary status of this special M to NC variable #25122 as 0 or 1.	R/W
MLC to NC Variable 124	M2x251	The system will move the binary status of this special M to NC variable #25123 as 0 or 1.	R/W
MLC to NC Variable 125	M2x252	The system will move the binary status of this special M to NC variable #25124 as 0 or 1.	R/W
MLC to NC Variable 126	M2x253	The system will move the binary status of this special M to NC variable #25125 as 0 or 1.	R/W
MLC to NC Variable 127	M2x254	The system will move the binary status of this special M to NC variable #25126 as 0 or 1.	R/W
MLC to NC Variable 128	M2x255	The system will move the binary status of this special M to NC variable #25127 as 0 or 1.	R/W
Synchronous Control Enable	M2x256	This special M must be set to ON when using synchronous functions to allow the NC to enable synchronous control.	R/W
Command Transfer Enable	M2x257	This special M must be set to ON when using command transfer functions to allow the NC to enable command transfer.	R/W

B

Function Name	Special M	Description	Device
Axes Oscillation Control Enable	M2x264	This special M must be set to ON when using axes oscillation functions. The system will execute axis oscillation commands based on the settings in **[D2x018 ~ D2x023]**. If the axis is already performing path interpolation, the oscillation command will be added to the original position command. Only available on modes other than HOME and EDIT.	R/W
X Axis Servo OFF	M2x272	Trigger this special M to set the X axis to Servo OFF status.	R/W
Y Axis Servo OFF	M2x273	Trigger this special M to set the Y axis to Servo OFF status.	R/W
Z Axis Servo OFF	M2x274	Trigger this special M to set the Z axis to Servo OFF status.	R/W
A Axis Servo OFF	M2x275	Trigger this special M to set the A axis to Servo OFF status.	R/W
B Axis Servo OFF	M2x276	Trigger this special M to set the B axis to Servo OFF status.	R/W
C Axis Servo OFF	M2x277	Trigger this special M to set the C axis to Servo OFF status.	R/W
U Axis Servo OFF	M2x278	Trigger this special M to set the U axis to Servo OFF status.	R/W
V Axis Servo OFF	M2x279	Trigger this special M to set the V axis to Servo OFF status.	R/W
W Axis Servo OFF	M2x280	Trigger this special M to set the W axis to Servo OFF status.	R/W
10th Axis Servo OFF	M2x281	Trigger this special M to set the 10th axis to Servo OFF status.	R/W
11th Axis Servo OFF	M2x282	Trigger this special M to set the 11th axis to Servo OFF status.	R/W
12th Axis Servo OFF	M2x283	Trigger this special M to set the 12th axis to Servo OFF status.	R/W
13th Axis Servo OFF	M2x284	Trigger this special M to set the 13th axis to Servo OFF status.	R/W
14th Axis Servo OFF	M2x285	Trigger this special M to set the 14th axis to Servo OFF status.	R/W
15th Axis Servo OFF	M2x286	Trigger this special M to set the 15th axis to Servo OFF status.	R/W
16th Axis Servo OFF	M2x287	Trigger this special M to set the 16th axis to Servo OFF status.	R/W
X Slave Axis Follows the Master Axis	M2x288	Sets the X axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
Y Slave Axis Follows the Master Axis	M2x289	Sets the Y axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
Z Slave Axis Follows the Master Axis	M2x290	Sets the Z axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
A Slave Axis Follows the Master Axis	M2x291	Sets the A axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
B Slave Axis Follows the Master Axis	M2x292	Sets the B axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
C Slave Axis Follows the Master Axis	M2x293	Sets the C axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
U Slave Axis Follows the Master Axis	M2x294	Sets the U axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
V Slave Axis Follows the Master Axis	M2x295	Sets the V axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
W Slave Axis Follows the Master Axis	M2x296	Sets the W axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W

Function Name	Special M	Description	Device
10th Slave Axis Follows the Master Axis	M2x297	Sets the 10th axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
11th Slave Axis Follows the Master Axis	M2x298	Sets the 11th axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
12th Slave Axis Follows the Master Axis	M2x299	Sets the 12th axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
13th Slave Axis Follows the Master Axis	M2x300	Sets the 13th axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
14th Slave Axis Follows the Master Axis	M2x301	Sets the 14th axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
15th Slave Axis Follows the Master Axis	M2x302	Sets the 15th axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
16th Slave Axis Follows the Master Axis	M2x303	Sets the 16th axis as the slave axis for synchronous control. The master axis ID must be set with **[N2.015]** in advance.	R/W
X Axis Receives Command from The Master Axis	M2x304	Specifies the X axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
Y Axis Receives Command from The Master Axis	M2x305	Specifies the Y axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
Z Axis Receives Command from The Master Axis	M2x306	Specifies the Z axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
A Axis Receives Command from The Master Axis	M2x307	Specifies the A axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
B Axis Receives Command from The Master Axis	M2x308	Specifies the B axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
C Axis Receives Command from The Master Axis	M2x309	Specifies the C axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
U Axis Receives Command from The Master Axis	M2x310	Specifies the U axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
V Axis Receives Command from The Master Axis	M2x311	Specifies the V axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
W Axis Receives Command from The Master Axis	M2x312	Specifies the W axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
10th Axis Receives Command from The Master Axis	M2x313	Specifies the 10th axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
11th Axis Receives Command from The Master Axis	M2x314	Specifies the 11th axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
12th Axis Receives Command from The Master Axis	M2x315	Specifies the 12th axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
13th Axis Receives Command from The Master Axis	M2x316	Specifies the 13th axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W

B

Function Name	Special M	Description	Device
14th Axis Receives Command from The Master Axis	M2x317	Specifies the 14th axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
15th Axis Receives Command from The Master Axis	M2x318	Specifies the 15th axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
16th Axis Receives Command from The Master Axis	M2x319	Specifies the 16th axis as the slave axis to receive the transfer command from the master axis. The master axis ID must be set with **[N2.015]** in advance.	R/W
X Axis Homing	M2x320	Trigger this special M for X axis homing procedure.	R/W
Y Axis Homing	M2x321	Trigger this special M for Y axis homing procedure.	R/W
Z Axis Homing	M2x322	Trigger this special M for Z axis homing procedure.	R/W
A Axis Homing	M2x323	Trigger this special M for A axis homing procedure.	R/W
B Axis Homing	M2x324	Trigger this special M for B axis homing procedure.	R/W
C Axis Homing	M2x325	Trigger this special M for C axis homing procedure.	R/W
U Axis Homing	M2x326	Trigger this special M for U axis homing procedure.	R/W
V Axis Homing	M2x327	Trigger this special M for V axis homing procedure.	R/W
W Axis Homing	M2x328	Trigger this special M for W axis homing procedure.	R/W
10th Axis Homing	M2x329	Trigger this special M for 10th axis homing procedure.	R/W
11th Axis Homing	M2x330	Trigger this special M for 11th axis homing procedure.	R/W
12th Axis Homing	M2x331	Trigger this special M for 12th axis homing procedure.	R/W
13th Axis Homing	M2x332	Trigger this special M for 13th axis homing procedure.	R/W
14th Axis Homing	M2x333	Trigger this special M for 14th axis homing procedure.	R/W
15th Axis Homing	M2x334	Trigger this special M for 15th axis homing procedure.	R/W
16th Axis Homing	M2x335	Trigger this special M for 16th axis homing procedure.	R/W
Lock X Axis Movement in Positive Direction	M2x336	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock Y Axis Movement in Positive Direction	M2x337	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock Z Axis Movement in Positive Direction	M2x338	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock A Axis Movement in Positive Direction	M2x339	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock B Axis Movement in Positive Direction	M2x340	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock C Axis Movement in Positive Direction	M2x341	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock U Axis Movement in Positive Direction	M2x342	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock V Axis Movement in Positive Direction	M2x343	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock W Axis Movement in Positive Direction	M2x344	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock 10th Axis Movement in Positive Direction	M2x345	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock 11th Axis Movement in Positive Direction	M2x346	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock 12th Axis Movement in Positive Direction	M2x347	When this special M is ON, the axis will not be able to move in the positive direction.	R/W

Function Name	Special M	Description	Device
Lock 13th Axis Movement in Positive Direction	M2x348	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock 14th Axis Movement in Positive Direction	M2x349	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock 15th Axis Movement in Positive Direction	M2x350	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock 16th Axis Movement in Positive Direction	M2x351	When this special M is ON, the axis will not be able to move in the positive direction.	R/W
Lock X Axis Movement in Negative Direction	M2x352	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock Y Axis Movement in Negative Direction	M2x353	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock Z Axis Movement in Negative Direction	M2x354	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock A Axis Movement in Negative Direction	M2x355	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock B Axis Movement in Negative Direction	M2x356	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock C Axis Movement in Negative Direction	M2x357	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock U Axis Movement in Negative Direction	M2x358	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock V Axis Movement in Negative Direction	M2x359	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock W Axis Movement in Negative Direction	M2x360	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock 10th Axis Movement in Negative Direction	M2x361	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock 11th Axis Movement in Negative Direction	M2x362	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock 12th Axis Movement in Negative Direction	M2x363	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock 13th Axis Movement in Negative Direction	M2x364	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock 14th Axis Movement in Negative Direction	M2x365	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock 15th Axis Movement in Negative Direction	M2x366	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Lock 16th Axis Movement in Negative Direction	M2x367	When this special M is ON, the axis will not be able to move in the negative direction.	R/W
Disable X Axis 1st Software Limit	M2x368	Trigger this special M to disable the X axis 1st software limit.	R/W
Disable Y Axis 1st Software Limit	M2x369	Trigger this special M to disable the Y axis 1st software limit.	R/W

B

Function Name	Special M	Description	Device
Disable Z Axis 1st Software Limit	M2x370	Trigger this special M to disable the Z axis 1st software limit.	R/W
Disable A Axis 1st Software Limit	M2x371	Trigger this special M to disable the A axis 1st software limit.	R/W
Disable B Axis 1st Software Limit	M2x372	Trigger this special M to disable the B axis 1st software limit.	R/W
Disable C Axis 1st Software Limit	M2x373	Trigger this special M to disable the C axis 1st software limit.	R/W
Disable U Axis 1st Software Limit	M2x374	Trigger this special M to disable the U axis 1st software limit.	R/W
Disable V Axis 1st Software Limit	M2x375	Trigger this special M to disable the V axis 1st software limit.	R/W
Disable W Axis 1st Software Limit	M2x376	Trigger this special M to disable the W axis 1st software limit.	R/W
Disable 10th Axis 1st Software Limit	M2x377	Trigger this special M to disable the 10th axis 1st software limit.	R/W
Disable 11th Axis 1st Software Limit	M2x378	Trigger this special M to disable the 11th axis 1st software limit.	R/W
Disable 12th Axis 1st Software Limit	M2x379	Trigger this special M to disable the 12th axis 1st software limit.	R/W
Disable 13th Axis 1st Software Limit	M2x380	Trigger this special M to disable the 13th axis 1st software limit.	R/W
Disable 14th Axis 1st Software Limit	M2x381	Trigger this special M to disable the 14th axis 1st software limit.	R/W
Disable 15th Axis 1st Software Limit	M2x382	Trigger this special M to disable the 15th axis 1st software limit.	R/W
Disable 16th Axis 1st Software Limit	M2x383	Trigger this special M to disable the 16th axis 1st software limit.	R/W
X Axis JOG Forward	M2x384	Trigger this special M for X axis forward JOG or INC operation.	R/W
Y Axis JOG Forward	M2x385	Trigger this special M for Y axis forward JOG or INC operation.	R/W
Z Axis JOG Forward	M2x386	Trigger this special M for Z axis forward JOG or INC operation.	R/W
A Axis JOG Forward	M2x387	Trigger this special M for A axis forward JOG or INC operation.	R/W
B Axis JOG Forward	M2x388	Trigger this special M for B axis forward JOG or INC operation.	R/W
C Axis JOG Forward	M2x389	Trigger this special M for C axis forward JOG or INC operation.	R/W
U Axis JOG Forward	M2x390	Trigger this special M for U axis forward JOG or INC operation.	R/W
V Axis JOG Forward	M2x391	Trigger this special M for V axis forward JOG or INC operation.	R/W
W Axis JOG Forward	M2x392	Trigger this special M for W axis forward JOG or INC operation.	R/W
10th Axis JOG Forward	M2x393	Trigger this special M for 10th axis forward JOG or INC operation.	R/W
11th Axis JOG Forward	M2x394	Trigger this special M for 11th axis forward JOG or INC operation.	R/W
12th Axis JOG Forward	M2x395	Trigger this special M for 12th axis forward JOG or INC operation.	R/W
13th Axis JOG Forward	M2x396	Trigger this special M for 13th axis forward JOG or INC operation.	R/W
14th Axis JOG Forward	M2x397	Trigger this special M for 14th axis forward JOG or INC operation.	R/W
15th Axis JOG Forward	M2x398	Trigger this special M for 15th axis forward JOG or INC operation.	R/W
16th Axis JOG Forward	M2x399	Trigger this special M for 16th axis forward JOG or INC operation.	R/W
X Axis JOG Reverse	M2x400	Trigger this special M for X axis reverse JOG or INC operation.	R/W

Function Name	Special M	Description	Device
Y Axis JOG Reverse	M2x401	Trigger this special M for Y axis reverse JOG or INC operation.	R/W
Z Axis JOG Reverse	M2x402	Trigger this special M for Z axis reverse JOG or INC operation.	R/W
A Axis JOG Reverse	M2x403	Trigger this special M for A axis reverse JOG or INC operation.	R/W
B Axis JOG Reverse	M2x404	Trigger this special M for B axis reverse JOG or INC operation.	R/W
C Axis JOG Reverse	M2x405	Trigger this special M for C axis reverse JOG or INC operation.	R/W
U Axis JOG Reverse	M2x406	Trigger this special M for U axis reverse JOG or INC operation.	R/W
V Axis JOG Reverse	M2x407	Trigger this special M for V axis reverse JOG or INC operation.	R/W
W Axis JOG Reverse	M2x408	Trigger this special M for W axis reverse JOG or INC operation.	R/W
10th Axis JOG Reverse	M2x409	Trigger this special M for 10th axis reverse JOG or INC operation.	R/W
11th Axis JOG Reverse	M2x410	Trigger this special M for 11th axis reverse JOG or INC operation.	R/W
12th Axis JOG Reverse	M2x411	Trigger this special M for 12th axis reverse JOG or INC operation.	R/W
13th Axis JOG Reverse	M2x412	Trigger this special M for 13th axis reverse JOG or INC operation.	R/W
14th Axis JOG Reverse	M2x413	Trigger this special M for 14th axis reverse JOG or INC operation.	R/W
15th Axis JOG Reverse	M2x414	Trigger this special M for 15th axis reverse JOG or INC operation.	R/W
16th Axis JOG Reverse	M2x415	Trigger this special M for 16th axis reverse JOG or INC operation.	R/W
MLC X Axis Control Mode	M2x416	When this special M is ON, X axis will be in speed mode. When this special M is OFF, X axis will be in position mode.	R/W
MLC Y Axis Control Mode	M2x417	When this special M is ON, Y axis will be in speed mode. When this special M is OFF, Y axis will be in position mode.	R/W
MLC Z Axis Control Mode	M2x418	When this special M is ON, Z axis will be in speed mode. When this special M is OFF, Z axis will be in position mode.	R/W
MLC A Axis Control Mode	M2x419	When this special M is ON, A axis will be in speed mode. When this special M is OFF, A axis will be in position mode.	R/W
MLC B Axis Control Mode	M2x420	When this special M is ON, B axis will be in speed mode. When this special M is OFF, B axis will be in position mode.	R/W
MLC C Axis Control Mode	M2x421	When this special M is ON, C axis will be in speed mode. When this special M is OFF, C axis will be in position mode.	R/W
MLC U Axis Control Mode	M2x422	When this special M is ON, U axis will be in speed mode. When this special M is OFF, U axis will be in position mode.	R/W
MLC V Axis Control Mode	M2x423	When this special M is ON, V axis will be in speed mode. When this special M is OFF, V axis will be in position mode.	R/W
MLC W Axis Control Mode	M2x424	When this special M is ON, W axis will be in speed mode. When this special M is OFF, W axis will be in position mode.	R/W
MLC 10th Axis Control Mode	M2x425	When this special M is ON, 10th axis will be in speed mode. When this special M is OFF, 10th axis will be in position mode.	R/W
MLC 11th Axis Control Mode	M2x426	When this special M is ON, 11th axis will be in speed mode. When this special M is OFF, 11th axis will be in position mode.	R/W

B

Function Name	Special M	Description	Device
MLC 12th Axis Control Mode	M2x427	When this special M is ON, 12th axis will be in speed mode. When this special M is OFF, 12th axis will be in position mode.	R/W
MLC 13th Axis Control Mode	M2x428	When this special M is ON, 13th axis will be in speed mode. When this special M is OFF, 13th axis will be in position mode.	R/W
MLC 14th Axis Control Mode	M2x429	When this special M is ON, 14th axis will be in speed mode. When this special M is OFF, 14th axis will be in position mode.	R/W
MLC 15th Axis Control Mode	M2x430	When this special M is ON, 15th axis will be in speed mode. When this special M is OFF, 15th axis will be in position mode.	R/W
MLC 16th Axis Control Mode	M2x431	When this special M is ON, 16th axis will be in speed mode. When this special M is OFF, 16th axis will be in position mode.	R/W
NC / MLC Axis Switching (X Axis)	M2x432	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (Y Axis)	M2x433	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (Z Axis)	M2x434	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (A Axis)	M2x435	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (B Axis)	M2x436	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (C Axis)	M2x437	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (U Axis)	M2x438	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (V Axis)	M2x439	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (W Axis)	M2x440	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (10th Axis)	M2x441	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (11th Axis)	M2x442	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (12th Axis)	M2x443	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (13th Axis)	M2x444	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (14th Axis)	M2x445	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (15th Axis)	M2x446	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W
NC / MLC Axis Switching (16th Axis)	M2x447	When this special M is ON, the axis will be controlled by MLC. When this special M is OFF, the axis will be controlled from NC command.	R/W

Function Name	Special M	Description	Device
Trigger X Axis' Movement (MLC Axis)	M2x448	Trigger this special M for MLC X axis movement.	R/W
Trigger Y Axis' Movement (MLC Axis)	M2x449	Trigger this special M for MLC Y axis movement.	R/W
Trigger Z Axis' Movement (MLC Axis)	M2x450	Trigger this special M for MLC Z axis movement.	R/W
Trigger A Axis' Movement (MLC Axis)	M2x451	Trigger this special M for MLC A axis movement.	R/W
Trigger B Axis' Movement (MLC Axis)	M2x452	Trigger this special M for MLC B axis movement.	R/W
Trigger C Axis' Movement (MLC Axis)	M2x453	Trigger this special M for MLC C axis movement.	R/W
Trigger U Axis' Movement (MLC Axis)	M2x454	Trigger this special M for MLC U axis movement.	R/W
Trigger V Axis' Movement (MLC Axis)	M2x455	Trigger this special M for MLC V axis movement.	R/W
Trigger W Axis' Movement (MLC Axis)	M2x456	Trigger this special M for MLC W axis movement.	R/W
Trigger 10th Axis' Movement (MLC Axis)	M2x457	Trigger this special M for MLC 10th axis movement.	R/W
Trigger 11th Axis' Movement (MLC Axis)	M2x458	Trigger this special M for MLC 11th axis movement.	R/W
Trigger 12th Axis' Movement (MLC Axis)	M2x459	Trigger this special M for MLC 12th axis movement.	R/W
Trigger 13th Axis' Movement (MLC Axis)	M2x460	Trigger this special M for MLC 13th axis movement.	R/W
Trigger 14th Axis' Movement (MLC Axis)	M2x461	Trigger this special M for MLC 14th axis movement.	R/W
Trigger 15th Axis' Movement (MLC Axis)	M2x462	Trigger this special M for MLC 15th axis movement.	R/W
Trigger 16th Axis' Movement (MLC Axis)	M2x463	Trigger this special M for MLC 16th axis movement.	R/W
MLC Axis Command Type of X Axis	M2x464	When this special M is ON, the **D2x256** will be incremental movement.	R/W
MLC Axis Command Type of Y Axis	M2x465	When this special M is ON, the **D2x258** will be incremental movement.	R/W
MLC Axis Command Type of Z Axis	M2x466	When this special M is ON, the **D2x260** will be incremental movement.	R/W
MLC Axis Command Type of A Axis	M2x467	When this special M is ON, the **D2x262** will be incremental movement.	R/W
MLC Axis Command Type of B Axis	M2x468	When this special M is ON, the **D2x264** will be incremental movement.	R/W

B

Function Name	Special M	Description	Device
MLC Axis Command Type of C Axis	M2x469	When this special M is ON, the **D2x266** will be incremental movement.	R/W
MLC Axis Command Type of U Axis	M2x470	When this special M is ON, the **D2x268** will be incremental movement.	R/W
MLC Axis Command Type of V Axis	M2x471	When this special M is ON, the **D2x270** will be incremental movement.	R/W
MLC Axis Command Type of W Axis	M2x472	When this special M is ON, the **D2x272** will be incremental movement.	R/W
MLC Axis Command Type of 10th Axis	M2x473	When this special M is ON, the **D2x274** will be incremental movement.	R/W
MLC Axis Command Type of 11th Axis	M2x474	When this special M is ON, the **D2x276** will be incremental movement.	R/W
MLC Axis Command Type of 12th Axis	M2x475	When this special M is ON, the **D2x278** will be incremental movement.	R/W
MLC Axis Command Type of 13th Axis	M2x476	When this special M is ON, the **D2x280** will be incremental movement.	R/W
MLC Axis Command Type of 14th Axis	M2x477	When this special M is ON, the **D2x282** will be incremental movement.	R/W
MLC Axis Command Type of 15th Axis	M2x478	When this special M is ON, the **D2x284** will be incremental movement.	R/W
MLC Axis Command Type of 16th Axis	M2x479	When this special M is ON, the **D2x286** will be incremental movement.	R/W
1st Spindle Forward Rotation	M2x704	Sets the 1st spindle to rotate in forward direction.	R/W
1st Spindle Reverse Rotation	M2x705	Sets the 1st spindle to rotate in reverse direction.	R/W
1st Spindle Positioning Control	M2x706	Positioning the 1st spindle.	R/W
1st Spindle Retraction After Tapping	M2x707	Retracts the 1st spindle after tapping.	R/W
Switching C / S Axis of 1st Spindle	M2x708	Trigger to switch between C or S axes for the spindle.	R/W
1st Spindle Command Source	M2x710	When this special M is ON, the speed command for the 1st spindle will refer to **D2x024**. When this special M is OFF, the speed command will be based on the S command from the NC program.	R/W
1st Spindle Alarm	M2x711	When the MLC receive an alarm signal from digital input and then trigger this special M, the system will stop NC process and return alarm code [0x0C0A MLC activate spindle alarm] of 1st spindle.	R/W
2nd Spindle Forward Rotation	M2x720	Sets the 2nd spindle to rotate in forward direction.	R/W
2nd Spindle Reverse Rotation	M2x721	Sets the 2nd spindle to rotate in reverse direction.	R/W
2nd Spindle Positioning Control	M2x722	Positioning the 2nd spindle.	R/W
2nd Spindle Retraction After Tapping	M2x723	Retracts the 2nd spindle after tapping.	R/W
Switching C / S Axis of 2nd Spindle	M2x724	Trigger to switch between C or S axes for the spindle.	R/W
2nd Spindle Command Source	M2x726	When this special M is ON, the speed command for the 2nd spindle will refer to **D2x030**. When this special M is OFF, the speed command will be based on the S command from the NC program.	R/W
2nd Spindle Alarm	M2x727	When the MLC receive an alarm signal from digital input and then trigger this special M, the system will stop NC process and return alarm code [0x0C0A MLC activate spindle alarm] of 2nd spindle.	R/W

Function Name	Special M	Description	Device
3rd Spindle Forward Rotation	M2x736	Sets the 3rd spindle to rotate in forward direction.	R/W
3rd Spindle Reverse Rotation	M2x737	Sets the 3rd spindle to rotate in reverse direction.	R/W
3rd Spindle Positioning Control	M2x738	Positioning the 3rd spindle.	R/W
3rd Spindle Retraction After Tapping	M2x739	Retracts the 3rd spindle after tapping.	R/W
Switching C / S Axis of 3rd Spindle	M2x740	Trigger to switch between C or S axes for the spindle.	R/W
3rd Spindle Command Source	M2x742	When this special M is ON, the speed command for the 3rd spindle will refer to **D2x320**. When this special M is OFF, the speed command will be based on the S command from the NC program.	R/W
3rd Spindle Alarm	M2x743	When the MLC receive an alarm signal from digital input and then trigger this special M, the system will stop NC process and return alarm code [0x0C0A MLC activate spindle alarm] of 3rd spindle.	R/W
4th Spindle Forward Rotation	M2x752	Sets the 4th spindle to rotate in forward direction.	R/W
4th Spindle Reverse Rotation	M2x753	Sets the 4th spindle to rotate in reverse direction.	R/W
4th Spindle Positioning Control	M2x754	Positioning the 4th spindle.	R/W
4th Spindle Retraction After Tapping	M2x755	Retracts the 4th spindle after tapping.	R/W
Switching C / S Axis of 4th Spindle	M2x756	Trigger to switch between C or S axes for the spindle.	R/W
4th Spindle Command Source	M2x758	When this special M is ON, the speed command for the 4th spindle will refer to **D2x326**. When this special M is OFF, the speed command will be based on the S command from the NC program.	R/W
4th Spindle Alarm	M2x759	When the MLC receive an alarm signal from digital input and then trigger this special M, the system will stop NC process and return alarm code [0x0C0A MLC activate spindle alarm] of 4th spindle.	R/W
5th Spindle Forward Rotation	M2x768	Sets the 5th spindle to rotate in forward direction.	R/W
5th Spindle Reverse Rotation	M2x769	Sets the 5th spindle to rotate in reverse direction.	R/W
5th Spindle Positioning Control	M2x770	Positioning the 5th spindle.	R/W
5th Spindle Retraction After Tapping	M2x771	Retracts the 5th spindle after tapping.	R/W
Switching C / S Axis of 5th Spindle	M2x772	Trigger to switch between C or S axes for the spindle.	R/W
5th Spindle Command Source	M2x774	When this special M is ON, the speed command for the 5th spindle will refer to **D2x332**. When this special M is OFF, the speed command will be based on the S command from the NC program.	R/W
5th Spindle Alarm	M2x775	When the MLC receive an alarm signal from digital input and then trigger this special M, the system will stop NC process and return alarm code [0x0C0A MLC activate spindle alarm] of 5th spindle.	R/W
6th Spindle Forward Rotation	M2x784	Sets the 6th spindle to rotate in forward direction.	R/W
6th Spindle Reverse Rotation	M2x785	Sets the 6th spindle to rotate in reverse direction.	R/W

B

Function Name	Special M	Description	Device
6th Spindle Positioning Control	M2x786	Positioning the 6th spindle.	R/W
6th Spindle Retraction After Tapping	M2x787	Retracts the 6th spindle after tapping.	R/W
Switching C / S Axis of 6th Spindle	M2x788	Trigger to switch between C or S axes for the spindle.	R/W
6th Spindle Command Source	M2x790	When this special M is ON, the speed command for the 6th spindle will refer to **D2x338**. When this special M is OFF, the speed command will be based on the S command from the NC program.	R/W
6th Spindle Alarm	M2x791	When the MLC receive an alarm signal from digital input and then trigger this special M, the system will stop NC process and return alarm code [0x0C0A MLC activate spindle alarm] of 6th spindle.	R/W
7th Spindle Forward Rotation	M2x800	Sets the 7th spindle to rotate in forward direction.	R/W
7th Spindle Reverse Rotation	M2x801	Sets the 7th spindle to rotate in reverse direction.	R/W
7th Spindle Positioning Control	M2x802	Positioning the 7th spindle.	R/W
7th Spindle Retraction After Tapping	M2x803	Retracts the 7th spindle after tapping.	R/W
Switching C / S Axis of 7th Spindle	M2x804	Trigger to switch between C or S axes for the spindle.	R/W
7th Spindle Command Source	M2x806	When this special M is ON, the speed command for the 7th spindle will refer to **D2x344**. When this special M is OFF, the speed command will be based on the S command from the NC program.	R/W
7th Spindle Alarm	M2x807	When the MLC receive an alarm signal from digital input and then trigger this special M, the system will stop NC process and return alarm code [0x0C0A MLC activate spindle alarm] of 7th spindle.	R/W
8th Spindle Forward Rotation	M2x816	Sets the 8th spindle to rotate in forward direction.	R/W
8th Spindle Reverse Rotation	M2x817	Sets the 8th spindle to rotate in reverse direction.	R/W
8th Spindle Positioning Control	M2x818	Positioning the 8th spindle.	R/W
8th Spindle Retraction After Tapping	M2x819	Retracts the 8th spindle after tapping.	R/W
Switching C / S Axis of 8th Spindle	M2x820	Trigger to switch between C or S axes for the spindle.	R/W
8th Spindle Command Source	M2x822	When this special M is ON, the speed command for the 8th spindle will refer to **D2x350**. When this special M is OFF, the speed command will be based on the S command from the NC program.	R/W
8th Spindle Alarm	M2x807	When the MLC receive an alarm signal from digital input and then trigger this special M, the system will stop NC process and return alarm code [0x0C0A MLC activate spindle alarm] of 8th spindle.	R/W
Time Pulse (500 ms)	M29032	This special M will continuously switch between ON and OFF for 500 ms each.	R
Time Pulse (50 ms)	M29033	This special M will continuously switch between ON and OFF for 50 ms each.	R
Remaining OFF	M29034	After the controller system is ready, this special M will always remain OFF.	R
Remaining ON	M29035	After the controller system is ready, this special M will always remain ON.	R

B.2.1.2 M49000 to M49899

Function Name	Special M	Description	Device
Program Lock	M49000	When this special M is ON, the system will lock the current main program and not allow changes or edits.	R/W

B.2.2 List of special M (System status)

B.2.2.1 M30000 to M38999

B

Function Name	Special M	Description	Device
EtherCAT Communication Status	M30000	After EtherCAT communication is initialized and without errors, this special M will be ON. Note: this special M status only corresponds to the EtherCAT communication status, it does not indicate Servo ON.	R
HSI 1 Status	M30016	When the system executes the skip command G31P1, this special M will be also set to ON by the NC.	R
HSI 2 Status	M30017	When the system executes the skip command G31P2, this special M will be also set to ON by the NC.	R
HSI 3 Status	M30018	When the system executes the skip command G31P3, this special M will be also set to ON by the NC.	R
HSI 4 Status	M30019	When the system executes the skip command G31P4, this special M will be also set to ON by the NC.	R
HSI 5 Status	M30020	When the system executes the skip command G31P5, this special M will be also set to ON by the NC.	R
HSI 6 Status	M30021	When the system executes the skip command G31P6, this special M will be also set to ON by the NC.	R
HSI 7 Status	M30022	When the system executes the skip command G31P7, this special M will be also set to ON by the NC.	R
HSI 8 Status	M30023	When the system executes the skip command G31P8, this special M will be also set to ON by the NC.	R
EIO 1 Connection Status	M30032	This special M is ON when the EtherCAT remote module node ID 501 is connected.	R
EIO 2 Connection Status	M30033	This special M is ON when the EtherCAT remote module node ID 502 is connected.	R
EIO 3 Connection Status	M30034	This special M is ON when the EtherCAT remote module node ID 503 is connected.	R
EIO 4 Connection Status	M30035	This special M is ON when the EtherCAT remote module node ID 504 is connected.	R
EIO 5 Connection Status	M30036	This special M is ON when the EtherCAT remote module node ID 505 is connected.	R
EIO 6 Connection Status	M30037	This special M is ON when the EtherCAT remote module node ID 506 is connected.	R
EIO 7 Connection Status	M30038	This special M is ON when the EtherCAT remote module node ID 507 is connected.	R
EIO 8 Connection Status	M30039	This special M is ON when the EtherCAT remote module node ID 508 is connected.	R
EIO 9 Connection Status	M30040	This special M is ON when the EtherCAT remote module node ID 509 is connected.	R
EIO 10 Connection Status	M30041	This special M is ON when the EtherCAT remote module node ID 510 is connected.	R
EIO 11 Connection Status	M30042	This special M is ON when the EtherCAT remote module node ID 511 is connected.	R
EIO 12 Connection Status	M30043	This special M is ON when the EtherCAT remote module node ID 512 is connected.	R
EIO 13 Connection Status	M30044	This special M is ON when the EtherCAT remote module node ID 513 is connected.	R
EIO 14 Connection Status	M30045	This special M is ON when the EtherCAT remote module node ID 514 is connected.	R
EIO 15 Connection Status	M30046	This special M is ON when the EtherCAT remote module node ID 515 is connected.	R
EIO 16 Connection Status	M30047	This special M is ON when the EtherCAT remote module node ID 516 is connected.	R
EIO 17 Connection Status	M30048	This special M is ON when the EtherCAT remote module node ID 517 is connected.	R

Function Name	Special M	Description	Device
EIO 18 Connection Status	M30049	This special M is ON when the EtherCAT remote module node ID 518 is connected.	R
EIO 19 Connection Status	M30050	This special M is ON when the EtherCAT remote module node ID 519 is connected.	R
EIO 20 Connection Status	M30051	This special M is ON when the EtherCAT remote module node ID 520 is connected.	R
NC Channel 1 Enable Status	M30080	This special M is ON when the 1st NC channel is enabled.	R
NC Channel 2 Enable Status	M30081	This special M is ON when the 2nd NC channel is enabled.	R
NC Channel 3 Enable Status	M30082	This special M is ON when the 3rd NC channel is enabled.	R
NC Channel 4 Enable Status	M30083	This special M is ON when the 4th NC channel is enabled.	R
Spindle 1 DI Status	M30096	This special M is normal close type relay, means it will always stay ON until Pin 8 on the Spindle 1 connector has turn ON, and this special M will be OFF instead.	R
Spindle 2 DI Status	M30097	This special M is normal close type relay, means it will always stay ON until Pin 8 on the Spindle 2 connector has turn ON, and this special M will be OFF instead.	R
G900 Command Control M_1	M30200	These Special M relay can be controlled by the G900 P_ Q_ command. This G900 will not stop look-ahead or pause the NC process. P: command value indicates the M address directly. From 30200 as M30200 to 30399 as M30399. When the value exceeds this range, the system will not return an error, but not the other special M will be triggered as well. Q: control the M relay status by 0 as OFF and any not 0 value will set as ON.	R
G900 Command Control M_2	M30201		R
G900 Command Control M_3	M30202		R
G900 Command Control M_4	M30203		R
G900 Command Control M_5	M30204		R
G900 Command Control M_6	M30205		R
G900 Command Control M_7	M30206		R
G900 Command Control M_8	M30207		R
G900 Command Control M_9	M30208		R
G900 Command Control M_10	M30209		R
G900 Command Control M_11	M30210		R
G900 Command Control M_12	M30211		R
G900 Command Control M_13	M30212		R
G900 Command Control M_14	M30213		R
G900 Command Control M_15	M30214		R
G900 Command Control M_16	M30215		R
G900 Command Control M_17	M30216		R
G900 Command Control M_18	M30217		R

B

Function Name	Special M	Description	Device
G900 Command Control M_19	M30218	These Special M relay can be controlled by the G900 P_ Q_ command. This G900 will not stop look-ahead or pause the NC process. P: command value indicates the M address directly. From 30200 as M30200 to 30399 as M30399. When the value exceeds this range, the system will not return an error, but not the other special M will be triggered as well. Q: control the M relay status by 0 as OFF and any not 0 value will set as ON.	R
G900 Command Control M_20	M30219		R
G900 Command Control M_21	M30220		R
G900 Command Control M_22	M30221		R
G900 Command Control M_23	M30222		R
G900 Command Control M_24	M30223		R
G900 Command Control M_25	M30224		R
G900 Command Control M_26	M30225		R
G900 Command Control M_27	M30226		R
G900 Command Control M_28	M30227		R
G900 Command Control M_29	M30228		R
G900 Command Control M_30	M30229		R
G900 Command Control M_31	M30230		R
G900 Command Control M_32	M30231		R
G900 Command Control M_33	M30232		R
G900 Command Control M_34	M30233		R
G900 Command Control M_35	M30234		R
G900 Command Control M_36	M30235		R
G900 Command Control M_37	M30236		R
G900 Command Control M_38	M30237		R
G900 Command Control M_39	M30238		R
G900 Command Control M_40	M30239		R
G900 Command Control M_41	M30240		R
G900 Command Control M_42	M30241		R
G900 Command Control M_43	M30242		R
G900 Command Control M_44	M30243		R
G900 Command Control M_45	M30244		R
G900 Command Control M_46	M30245		R

Function Name	Special M	Description	Device
G900 Command Control M_47	M30246	These Special M relay can be controlled by the G900 P_ Q_ command. This G900 will not stop look-ahead or pause the NC process. P: command value indicates the M address directly. From 30200 as M30200 to 30399 as M30399. When the value exceeds this range, the system will not return an error, but not the other special M will be triggered as well. Q: control the M relay status by 0 as OFF and any not 0 value will set as ON.	R
G900 Command Control M_48	M30247		R
G900 Command Control M_49	M30248		R
G900 Command Control M_50	M30249		R
G900 Command Control M_51	M30250		R
G900 Command Control M_52	M30251		R
G900 Command Control M_53	M30252		R
G900 Command Control M_54	M30253		R
G900 Command Control M_55	M30254		R
G900 Command Control M_56	M30255		R
G900 Command Control M_57	M30256		R
G900 Command Control M_58	M30257		R
G900 Command Control M_59	M30258		R
G900 Command Control M_60	M30259		R
G900 Command Control M_61	M30260		R
G900 Command Control M_62	M30261		R
G900 Command Control M_63	M30262		R
G900 Command Control M_64	M30263		R
G900 Command Control M_65	M30264		R
G900 Command Control M_66	M30265		R
G900 Command Control M_67	M30266		R
G900 Command Control M_68	M30267		R
G900 Command Control M_69	M30268		R
G900 Command Control M_70	M30269		R
G900 Command Control M_71	M30270		R
G900 Command Control M_72	M30271		R
G900 Command Control M_73	M30272		R
G900 Command Control M_74	M30273		R

B

Function Name	Special M	Description	Device
G900 Command Control M_75	M30274	These Special M relay can be controlled by the G900 P_ Q_ command. This G900 will not stop look-ahead or pause the NC process. P: command value indicates the M address directly. From 30200 as M30200 to 30399 as M30399. When the value exceeds this range, the system will not return an error, but not the other special M will be triggered as well. Q: control the M relay status by 0 as OFF and any not 0 value will set as ON.	R
G900 Command Control M_76	M30275		R
G900 Command Control M_77	M30276		R
G900 Command Control M_78	M30277		R
G900 Command Control M_79	M30278		R
G900 Command Control M_80	M30279		R
G900 Command Control M_81	M30280		R
G900 Command Control M_82	M30281		R
G900 Command Control M_83	M30282		R
G900 Command Control M_84	M30283		R
G900 Command Control M_85	M30284		R
G900 Command Control M_86	M30285		R
G900 Command Control M_87	M30286		R
G900 Command Control M_88	M30287		R
G900 Command Control M_89	M30288		R
G900 Command Control M_90	M30289		R
G900 Command Control M_91	M30290		R
G900 Command Control M_92	M30291		R
G900 Command Control M_93	M30292		R
G900 Command Control M_94	M30293		R
G900 Command Control M_95	M30294		R
G900 Command Control M_96	M30295		R
G900 Command Control M_97	M30296		R
G900 Command Control M_98	M30297		R
G900 Command Control M_99	M30298		R
G900 Command Control M_100	M30299		R
G900 Command Control M_101	M30300		R
G900 Command Control M_102	M30301		R

Function Name	Special M	Description	Device
G900 Command Control M_103	M30302	These Special M relay can be controlled by the G900 P_ Q_ command. This G900 will not stop look-ahead or pause the NC process. P: command value indicates the M address directly. From 30200 as M30200 to 30399 as M30399. When the value exceeds this range, the system will not return an error, but not the other special M will be triggered as well. Q: control the M relay status by 0 as OFF and any not 0 value will set as ON.	R
G900 Command Control M_104	M30303		R
G900 Command Control M_105	M30304		R
G900 Command Control M_106	M30305		R
G900 Command Control M_107	M30306		R
G900 Command Control M_108	M30307		R
G900 Command Control M_109	M30308		R
G900 Command Control M_110	M30309		R
G900 Command Control M_111	M30310		R
G900 Command Control M_112	M30311		R
G900 Command Control M_113	M30312		R
G900 Command Control M_114	M30313		R
G900 Command Control M_115	M30314		R
G900 Command Control M_116	M30315		R
G900 Command Control M_117	M30316		R
G900 Command Control M_118	M30317		R
G900 Command Control M_119	M30318		R
G900 Command Control M_120	M30319		R
G900 Command Control M_121	M30320		R
G900 Command Control M_122	M30321		R
G900 Command Control M_123	M30322		R
G900 Command Control M_124	M30323		R
G900 Command Control M_125	M30324		R
G900 Command Control M_126	M30325		R
G900 Command Control M_127	M30326		R
G900 Command Control M_128	M30327		R
G900 Command Control M_129	M30328		R
G900 Command Control M_130	M30329		R

B

Function Name	Special M	Description	Device
G900 Command Control M_131	M30330	These Special M relay can be controlled by the G900 P_ Q_ command. This G900 will not stop look-ahead or pause the NC process. P: command value indicates the M address directly. From 30200 as M30200 to 30399 as M30399. When the value exceeds this range, the system will not return an error, but not the other special M will be triggered as well. Q: control the M relay status by 0 as OFF and any not 0 value will set as ON.	R
G900 Command Control M_132	M30331		R
G900 Command Control M_133	M30332		R
G900 Command Control M_134	M30333		R
G900 Command Control M_135	M30334		R
G900 Command Control M_136	M30335		R
G900 Command Control M_137	M30336		R
G900 Command Control M_138	M30337		R
G900 Command Control M_139	M30338		R
G900 Command Control M_140	M30339		R
G900 Command Control M_141	M30340		R
G900 Command Control M_142	M30341		R
G900 Command Control M_143	M30342		R
G900 Command Control M_144	M30343		R
G900 Command Control M_145	M30344		R
G900 Command Control M_146	M30345		R
G900 Command Control M_147	M30346		R
G900 Command Control M_148	M30347		R
G900 Command Control M_149	M30348		R
G900 Command Control M_150	M30349		R
G900 Command Control M_151	M30350		R
G900 Command Control M_152	M30351		R
G900 Command Control M_153	M30352		R
G900 Command Control M_154	M30353		R
G900 Command Control M_155	M30354		R
G900 Command Control M_156	M30355		R
G900 Command Control M_157	M30356		R
G900 Command Control M_158	M30357		R

Function Name	Special M	Description	Device
G900 Command Control M_159	M30358	These Special M relay can be controlled by the G900 P_ Q_ command. This G900 will not stop look-ahead or pause the NC process. P: command value indicates the M address directly. From 30200 as M30200 to 30399 as M30399. When the value exceeds this range, the system will not return an error, but not the other special M will be triggered as well. Q: control the M relay status by 0 as OFF and any not 0 value will set as ON.	R
G900 Command Control M_160	M30359		R
G900 Command Control M_161	M30360		R
G900 Command Control M_162	M30361		R
G900 Command Control M_163	M30362		R
G900 Command Control M_164	M30363		R
G900 Command Control M_165	M30364		R
G900 Command Control M_166	M30365		R
G900 Command Control M_167	M30366		R
G900 Command Control M_168	M30367		R
G900 Command Control M_169	M30368		R
G900 Command Control M_170	M30369		R
G900 Command Control M_171	M30370		R
G900 Command Control M_172	M30371		R
G900 Command Control M_173	M30372		R
G900 Command Control M_174	M30373		R
G900 Command Control M_175	M30374		R
G900 Command Control M_176	M30375		R
G900 Command Control M_177	M30376		R
G900 Command Control M_178	M30377		R
G900 Command Control M_179	M30378		R
G900 Command Control M_180	M30379		R
G900 Command Control M_181	M30380		R
G900 Command Control M_182	M30381		R
G900 Command Control M_183	M30382		R
G900 Command Control M_184	M30383		R
G900 Command Control M_185	M30384		R
G900 Command Control M_186	M30385		R

B

Function Name	Special M	Description	Device
G900 Command Control M_187	M30386	These Special M relay can be controlled by the G900 P_ Q_ command. This G900 will not stop look-ahead or pause the NC process. P: command value indicates the M address directly. From 30200 as M30200 to 30399 as M30399. When the value exceeds this range, the system will not return an error, but not the other special M will be triggered as well. Q: control the M relay status by 0 as OFF and any not 0 value will set as ON.	R
G900 Command Control M_188	M30387		R
G900 Command Control M_189	M30388		R
G900 Command Control M_190	M30389		R
G900 Command Control M_191	M30390		R
G900 Command Control M_192	M30391		R
G900 Command Control M_193	M30392		R
G900 Command Control M_194	M30393		R
G900 Command Control M_195	M30394		R
G900 Command Control M_196	M30395		R
G900 Command Control M_197	M30396		R
G900 Command Control M_198	M30397		R
G900 Command Control M_199	M30398		R
G900 Command Control M_200	M30399		R
AUTO	M3x000	This special M is ON when the NC system is in AUTO mode.	R
EDIT	M3x001	This special M is ON when the NC system is in EDIT mode.	R
MDI	M3x002	This special M is ON when the NC system is in MDI mode.	R
MPG	M3x003	This special M is ON when the NC system is in MPG mode.	R
JOG	M3x004	This special M is ON when the NC system is in JOG mode.	R
RAPID	M3x005	This special M is ON when the NC system is in RAPID mode.	R
INC	M3x006	This special M is ON when the NC system is in INC mode.	R
HOME	M3x007	This special M is ON when the NC system is in HOME mode.	R
Cycle Start Status	M3x016	This special M is ON when the NC system is running NC program.	R
Feed Hold Status	M3x017	This special M is ON when the NC system pause NC process.	R
Emergency Stop Status	M3x018	This special M is ON when the EMG button or signal is triggered.	R
Reset Finished	M3x019	This special M is ON when the NC system finishes reset procedures.	R
Break Point Searching	M3x020	This special M is ON when the NC system is searching for the break point.	R
Program End Finished	M3x021	This special M is ON when the NC system finishes the last block, M02 or M30.	R
M02 Executed	M3x022	This special M is ON when the NC system finishes executing the M02 command and cycle stop.	R
M30 Cycle Stop and Index Reset	M3x023	This special M is ON when the NC system finishes executing the M30 command, cycle stop and program line index back to first line.	R

Function Name	Special M	Description	Device
Single Block Hold	M3x024	This special M is ON when the NC system is holding on a single block.	R
NC Error	M3x025	This special M is ON when the NC system encounters an error.	R
Macro Call Status	M3x027	This special M is ON when the macro call is in execution.	R
Macro Call Ready	M3x028	After users trigger the macro call preparation and then system finishes the preparation, this special M will be ON. When this special M is ON, users will need to switch the NC system to AUTO mode and finish the remaining actions to start the macro.	R
Macro Call Error	M3x029	Indicates a macro call error.	R
M96 (Program Interruption) in Execution	M3x031	This special M is ON when M96 (program interruption) is in execution.	R
System Mode Switching	M3x032	When the system is switching between operation modes such as AUTO or JOG, this special M will be triggered.	R
Main Program Lock	M3x033	When the system locks the current NC main program and it is not allowed to change, this special M will be triggered.	R
Servo Drive Error	M3x034	When any of the connected servo drives encounter an error, this special M will be triggered.	R
Axes Auto Servo ON Status	M3x035	When the system parameter **[N1.10 Axes Manual Servo ON]** is set to 0, the system will set all axes and this special M to servo ON after initialization is finished.	R
System Ready and Servo ON	M3x036	When the system is successful initialized and servo drives are set to servo ON, this special M will be triggered.	R
M00 System Hold	M3x037	After the system executes M00 and pauses the procedure, this special M will be triggered.	R
M01 Optional Stop	M3x038	After the system executes M01 and stops the procedure, this special M will be triggered.	R
Block Finished on Single Block Mode	M3x039	When the system is in single block mode and then finishes the current block, this special M will be triggered.	R
User Define Alarm Status	M3x040	This special M is ON when a user defined alarm occurs.	R
NC Program Scanning	M3x044	This special M is ON when the system is scanning the NC program.	R
NC Program Scan finished	M3x045	This special M is ON when the system finishes the NC program scanning.	R
M99 Call Stop Status	M3x047	When the **M2x026** is ON and execute the M99, the NC system will trigger this special M after slowing down the motion and stop the process. This special M will be reset to OFF when the cycle start triggered again, M30 executed or system reset.	R/W
1st Macro Call Initial Finished	M3x048	This special M is ON when macro call finishes.	R
2nd Macro Call Initial Finished	M3x049	This special M is ON when macro call finishes.	R
3rd Macro Call Initial Finished	M3x050	This special M is ON when macro call finishes.	R
4th Macro Call Initial Finished	M3x051	This special M is ON when macro call finishes.	R
5th Macro Call Initial Finished	M3x052	This special M is ON when macro call finishes.	R
6th Macro Call Initial Finished	M3x053	This special M is ON when macro call finishes.	R
7th Macro Call Initial Finished	M3x054	This special M is ON when macro call finishes.	R

B

Function Name	Special M	Description	Device
8th Macro Call Initial Finished	M3x055	This special M is ON when macro call finishes.	R
9th Macro Call Initial Finished	M3x056	This special M is ON when macro call finishes.	R
10th Macro Call Initial Finished	M3x057	This special M is ON when macro call finishes.	R
11th Macro Call Initial Finished	M3x058	This special M is ON when macro call finishes.	R
12th Macro Call Initial Finished	M3x059	This special M is ON when macro call finishes.	R
13th Macro Call Initial Finished	M3x060	This special M is ON when macro call finishes.	R
14th Macro Call Initial Finished	M3x061	This special M is ON when macro call finishes.	R
15th Macro Call Initial Finished	M3x062	This special M is ON when macro call finishes.	R
16th Macro Call Initial Finished	M3x063	This special M is ON when macro call finishes.	R
M Code Execution	M3x064	When the M code is executed in the program (not including M00, M01, M02, M30, M98, M99), this special M will be triggered. When the M, S, and T codes complete their execution and then the MLC triggers the **M2x016**, this special M will be set to OFF. This action does not include an M code that is used for macro calls.	R
S Code Execution	M3x065	When the S code is executed in the program, this special M will be triggered. When the M, S, and T codes complete their execution and then the MLC triggers the **M2x016**, this special M will be set to OFF. The NC does not trigger this special M when an S code is used for macro call.	R
T Code Execution	M3x066	When the T code (Standby tool number) is executed in the program, this special M will be triggered. When the M, S, and T codes complete their execution and then the MLC triggers the **M2x016**, this special M will be set to OFF. The NC does not trigger this special M when a T code is used for macro call. This special M is related to the tool pot setting of the tool magazine, and the special M will be triggered only when the T code value is set within the specified range of tool number for the tool magazine parameter.	R
Connection Status of 1st Spindle	M3x096	This special M is ON when the 1st spindle is connected.	R
Connection Status of 2nd Spindle	M3x097	This special M is ON when the 2nd spindle is connected.	R
Connection Status of 3rd Spindle	M3x098	This special M is ON when the 3rd spindle is connected.	R
Connection Status of 4th Spindle	M3x099	This special M is ON when the 4th spindle is connected.	R
Connection Status of 5th Spindle	M3x100	This special M is ON when the 5th spindle is connected.	R
Connection Status of 6th Spindle	M3x101	This special M is ON when the 6th spindle is connected.	R
Connection Status of 7th Spindle	M3x102	This special M is ON when the 7th spindle is connected.	R
Connection Status of 8th Spindle	M3x103	This special M is ON when the 8th spindle is connected.	R
NC Variable to MLC 1	M3x128	The NC system will update the status of variable #25256 into this special M as ON or OFF.	R
NC Variable to MLC 2	M3x129	The NC system will update the status of variable #25257 into this special M as ON or OFF.	R

Function Name	Special M	Description	Device
NC Variable to MLC 3	M3x130	The NC system will update the status of variable #25258 into this special M as ON or OFF.	R
NC Variable to MLC 4	M3x131	The NC system will update the status of variable #25259 into this special M as ON or OFF.	R
NC Variable to MLC 5	M3x132	The NC system will update the status of variable #25260 into this special M as ON or OFF.	R
NC Variable to MLC 6	M3x133	The NC system will update the status of variable #25261 into this special M as ON or OFF.	R
NC Variable to MLC 7	M3x134	The NC system will update the status of variable #25262 into this special M as ON or OFF.	R
NC Variable to MLC 8	M3x135	The NC system will update the status of variable #25263 into this special M as ON or OFF.	R
NC Variable to MLC 9	M3x136	The NC system will update the status of variable #25264 into this special M as ON or OFF.	R
NC Variable to MLC 10	M3x137	The NC system will update the status of variable #25265 into this special M as ON or OFF.	R
NC Variable to MLC 11	M3x138	The NC system will update the status of variable #25266 into this special M as ON or OFF.	R
NC Variable to MLC 12	M3x139	The NC system will update the status of variable #25267 into this special M as ON or OFF.	R
NC Variable to MLC 13	M3x140	The NC system will update the status of variable #25268 into this special M as ON or OFF.	R
NC Variable to MLC 14	M3x141	The NC system will update the status of variable #25269 into this special M as ON or OFF.	R
NC Variable to MLC 15	M3x142	The NC system will update the status of variable #25270 into this special M as ON or OFF.	R
NC Variable to MLC 16	M3x143	The NC system will update the status of variable #25271 into this special M as ON or OFF.	R
NC Variable to MLC 17	M3x144	The NC system will update the status of variable #25272 into this special M as ON or OFF.	R
NC Variable to MLC 18	M3x145	The NC system will update the status of variable #25273 into this special M as ON or OFF.	R
NC Variable to MLC 19	M3x146	The NC system will update the status of variable #25274 into this special M as ON or OFF.	R
NC Variable to MLC 20	M3x147	The NC system will update the status of variable #25275 into this special M as ON or OFF.	R
NC Variable to MLC 21	M3x148	The NC system will update the status of variable #25276 into this special M as ON or OFF.	R
NC Variable to MLC 22	M3x149	The NC system will update the status of variable #25277 into this special M as ON or OFF.	R
NC Variable to MLC 23	M3x150	The NC system will update the status of variable #25278 into this special M as ON or OFF.	R
NC Variable to MLC 24	M3x151	The NC system will update the status of variable #25279 into this special M as ON or OFF.	R
NC Variable to MLC 25	M3x152	The NC system will update the status of variable #25280 into this special M as ON or OFF.	R
NC Variable to MLC 26	M3x153	The NC system will update the status of variable #25281 into this special M as ON or OFF.	R
NC Variable to MLC 27	M3x154	The NC system will update the status of variable #25282 into this special M as ON or OFF.	R
NC Variable to MLC 28	M3x155	The NC system will update the status of variable #25283 into this special M as ON or OFF.	R
NC Variable to MLC 29	M3x156	The NC system will update the status of variable #25284 into this special M as ON or OFF.	R
NC Variable to MLC 30	M3x157	The NC system will update the status of variable #25285 into this special M as ON or OFF.	R
NC Variable to MLC 31	M3x158	The NC system will update the status of variable #25286 into this special M as ON or OFF.	R
NC Variable to MLC 32	M3x159	The NC system will update the status of variable #25287 into this special M as ON or OFF.	R
NC Variable to MLC 33	M3x160	The NC system will update the status of variable #25288 into this special M as ON or OFF.	R
NC Variable to MLC 34	M3x161	The NC system will update the status of variable #25289 into this special M as ON or OFF.	R

B

Function Name	Special M	Description	Device
NC Variable to MLC 35	M3x162	The NC system will update the status of variable #25290 into this special M as ON or OFF.	R
NC Variable to MLC 36	M3x163	The NC system will update the status of variable #25291 into this special M as ON or OFF.	R
NC Variable to MLC 37	M3x164	The NC system will update the status of variable #25292 into this special M as ON or OFF.	R
NC Variable to MLC 38	M3x165	The NC system will update the status of variable #25293 into this special M as ON or OFF.	R
NC Variable to MLC 39	M3x166	The NC system will update the status of variable #25294 into this special M as ON or OFF.	R
NC Variable to MLC 40	M3x167	The NC system will update the status of variable #25295 into this special M as ON or OFF.	R
NC Variable to MLC 41	M3x168	The NC system will update the status of variable #25296 into this special M as ON or OFF.	R
NC Variable to MLC 42	M3x169	The NC system will update the status of variable #25297 into this special M as ON or OFF.	R
NC Variable to MLC 43	M3x170	The NC system will update the status of variable #25298 into this special M as ON or OFF.	R
NC Variable to MLC 44	M3x171	The NC system will update the status of variable #25299 into this special M as ON or OFF.	R
NC Variable to MLC 45	M3x172	The NC system will update the status of variable #25300 into this special M as ON or OFF.	R
NC Variable to MLC 46	M3x173	The NC system will update the status of variable #25301 into this special M as ON or OFF.	R
NC Variable to MLC 47	M3x174	The NC system will update the status of variable #25302 into this special M as ON or OFF.	R
NC Variable to MLC 48	M3x175	The NC system will update the status of variable #25303 into this special M as ON or OFF.	R
NC Variable to MLC 49	M3x176	The NC system will update the status of variable #25304 into this special M as ON or OFF.	R
NC Variable to MLC 50	M3x177	The NC system will update the status of variable #25305 into this special M as ON or OFF.	R
NC Variable to MLC 51	M3x178	The NC system will update the status of variable #25306 into this special M as ON or OFF.	R
NC Variable to MLC 52	M3x179	The NC system will update the status of variable #25307 into this special M as ON or OFF.	R
NC Variable to MLC 53	M3x180	The NC system will update the status of variable #25308 into this special M as ON or OFF.	R
NC Variable to MLC 54	M3x181	The NC system will update the status of variable #25309 into this special M as ON or OFF.	R
NC Variable to MLC 55	M3x182	The NC system will update the status of variable #25310 into this special M as ON or OFF.	R
NC Variable to MLC 56	M3x183	The NC system will update the status of variable #25311 into this special M as ON or OFF.	R
NC Variable to MLC 57	M3x184	The NC system will update the status of variable #25312 into this special M as ON or OFF.	R
NC Variable to MLC 58	M3x185	The NC system will update the status of variable #25313 into this special M as ON or OFF.	R
NC Variable to MLC 59	M3x186	The NC system will update the status of variable #25314 into this special M as ON or OFF.	R
NC Variable to MLC 60	M3x187	The NC system will update the status of variable #25315 into this special M as ON or OFF.	R
NC Variable to MLC 61	M3x188	The NC system will update the status of variable #25316 into this special M as ON or OFF.	R
NC Variable to MLC 62	M3x189	The NC system will update the status of variable #25317 into this special M as ON or OFF.	R
NC Variable to MLC 63	M3x190	The NC system will update the status of variable #25318 into this special M as ON or OFF.	R
NC Variable to MLC 64	M3x191	The NC system will update the status of variable #25319 into this special M as ON or OFF.	R
NC Variable to MLC 65	M3x192	The NC system will update the status of variable #25320 into this special M as ON or OFF.	R
NC Variable to MLC 66	M3x193	The NC system will update the status of variable #25321 into this special M as ON or OFF.	R
NC Variable to MLC 67	M3x194	The NC system will update the status of variable #25322 into this special M as ON or OFF.	R

Function Name	Special M	Description	Device
NC Variable to MLC 68	M3x195	The NC system will update the status of variable #25323 into this special M as ON or OFF.	R
NC Variable to MLC 69	M3x196	The NC system will update the status of variable #25324 into this special M as ON or OFF.	R
NC Variable to MLC 70	M3x197	The NC system will update the status of variable #25325 into this special M as ON or OFF.	R
NC Variable to MLC 71	M3x198	The NC system will update the status of variable #25326 into this special M as ON or OFF.	R
NC Variable to MLC 72	M3x199	The NC system will update the status of variable #25327 into this special M as ON or OFF.	R
NC Variable to MLC 73	M3x200	The NC system will update the status of variable #25328 into this special M as ON or OFF.	R
NC Variable to MLC 74	M3x201	The NC system will update the status of variable #25329 into this special M as ON or OFF.	R
NC Variable to MLC 75	M3x202	The NC system will update the status of variable #25330 into this special M as ON or OFF.	R
NC Variable to MLC 76	M3x203	The NC system will update the status of variable #25331 into this special M as ON or OFF.	R
NC Variable to MLC 77	M3x204	The NC system will update the status of variable #25332 into this special M as ON or OFF.	R
NC Variable to MLC 78	M3x205	The NC system will update the status of variable #25333 into this special M as ON or OFF.	R
NC Variable to MLC 79	M3x206	The NC system will update the status of variable #25334 into this special M as ON or OFF.	R
NC Variable to MLC 80	M3x207	The NC system will update the status of variable #25335 into this special M as ON or OFF.	R
NC Variable to MLC 81	M3x208	The NC system will update the status of variable #25336 into this special M as ON or OFF.	R
NC Variable to MLC 82	M3x209	The NC system will update the status of variable #25337 into this special M as ON or OFF.	R
NC Variable to MLC 83	M3x210	The NC system will update the status of variable #25338 into this special M as ON or OFF.	R
NC Variable to MLC 84	M3x211	The NC system will update the status of variable #25339 into this special M as ON or OFF.	R
NC Variable to MLC 85	M3x212	The NC system will update the status of variable #25340 into this special M as ON or OFF.	R
NC Variable to MLC 86	M3x213	The NC system will update the status of variable #25341 into this special M as ON or OFF.	R
NC Variable to MLC 87	M3x214	The NC system will update the status of variable #25342 into this special M as ON or OFF.	R
NC Variable to MLC 88	M3x215	The NC system will update the status of variable #25343 into this special M as ON or OFF.	R
NC Variable to MLC 89	M3x216	The NC system will update the status of variable #25344 into this special M as ON or OFF.	R
NC Variable to MLC 90	M3x217	The NC system will update the status of variable #25345 into this special M as ON or OFF.	R
NC Variable to MLC 91	M3x218	The NC system will update the status of variable #25346 into this special M as ON or OFF.	R
NC Variable to MLC 92	M3x219	The NC system will update the status of variable #25347 into this special M as ON or OFF.	R
NC Variable to MLC 93	M3x220	The NC system will update the status of variable #25348 into this special M as ON or OFF.	R
NC Variable to MLC 94	M3x221	The NC system will update the status of variable #25349 into this special M as ON or OFF.	R
NC Variable to MLC 95	M3x222	The NC system will update the status of variable #25350 into this special M as ON or OFF.	R
NC Variable to MLC 96	M3x223	The NC system will update the status of variable #25351 into this special M as ON or OFF.	R
NC Variable to MLC 97	M3x224	The NC system will update the status of variable #25352 into this special M as ON or OFF.	R
NC Variable to MLC 98	M3x225	The NC system will update the status of variable #25353 into this special M as ON or OFF.	R
NC Variable to MLC 99	M3x226	The NC system will update the status of variable #25354 into this special M as ON or OFF.	R

B

Function Name	Special M	Description	Device
NC Variable to MLC 100	M3x227	The NC system will update the status of variable #25355 into this special M as ON or OFF.	R
NC Variable to MLC 101	M3x228	The NC system will update the status of variable #25356 into this special M as ON or OFF.	R
NC Variable to MLC 102	M3x229	The NC system will update the status of variable #25357 into this special M as ON or OFF.	R
NC Variable to MLC 103	M3x230	The NC system will update the status of variable #25358 into this special M as ON or OFF.	R
NC Variable to MLC 104	M3x231	The NC system will update the status of variable #25359 into this special M as ON or OFF.	R
NC Variable to MLC 105	M3x232	The NC system will update the status of variable #25360 into this special M as ON or OFF.	R
NC Variable to MLC 106	M3x233	The NC system will update the status of variable #25361 into this special M as ON or OFF.	R
NC Variable to MLC 107	M3x234	The NC system will update the status of variable #25362 into this special M as ON or OFF.	R
NC Variable to MLC 108	M3x235	The NC system will update the status of variable #25363 into this special M as ON or OFF.	R
NC Variable to MLC 109	M3x236	The NC system will update the status of variable #25364 into this special M as ON or OFF.	R
NC Variable to MLC 110	M3x237	The NC system will update the status of variable #25365 into this special M as ON or OFF.	R
NC Variable to MLC 111	M3x238	The NC system will update the status of variable #25366 into this special M as ON or OFF.	R
NC Variable to MLC 112	M3x239	The NC system will update the status of variable #25367 into this special M as ON or OFF.	R
NC Variable to MLC 113	M3x240	The NC system will update the status of variable #25368 into this special M as ON or OFF.	R
NC Variable to MLC 114	M3x241	The NC system will update the status of variable #25369 into this special M as ON or OFF.	R
NC Variable to MLC 115	M3x242	The NC system will update the status of variable #25370 into this special M as ON or OFF.	R
NC Variable to MLC 116	M3x243	The NC system will update the status of variable #25371 into this special M as ON or OFF.	R
NC Variable to MLC 117	M3x244	The NC system will update the status of variable #25372 into this special M as ON or OFF.	R
NC Variable to MLC 118	M3x245	The NC system will update the status of variable #25373 into this special M as ON or OFF.	R
NC Variable to MLC 119	M3x246	The NC system will update the status of variable #25374 into this special M as ON or OFF.	R
NC Variable to MLC 120	M3x247	The NC system will update the status of variable #25375 into this special M as ON or OFF.	R
NC Variable to MLC 121	M3x248	The NC system will update the status of variable #25376 into this special M as ON or OFF.	R
NC Variable to MLC 122	M3x249	The NC system will update the status of variable #25377 into this special M as ON or OFF.	R
NC Variable to MLC 123	M3x250	The NC system will update the status of variable #25378 into this special M as ON or OFF.	R
NC Variable to MLC 124	M3x251	The NC system will update the status of variable #25379 into this special M as ON or OFF.	R
NC Variable to MLC 125	M3x252	The NC system will update the status of variable #25380 into this special M as ON or OFF.	R
NC Variable to MLC 126	M3x253	The NC system will update the status of variable #25381 into this special M as ON or OFF.	R
NC Variable to MLC 127	M3x254	The NC system will update the status of variable #25382 into this special M as ON or OFF.	R

Function Name	Special M	Description	Device
NC Variable to MLC 128	M3x255	The NC system will update the status of variable #25383 into this special M as ON or OFF.	R
Servo Connection Status of X Axis	M3x256	This special M is ON when the servo drive of the X axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of Y Axis	M3x257	This special M is ON when the servo drive of the Y axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of Z Axis	M3x258	This special M is ON when the servo drive of the Z axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of A Axis	M3x259	This special M is ON when the servo drive of the A axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of B Axis	M3x260	This special M is ON when the servo drive of the B axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of C Axis	M3x261	This special M is ON when the servo drive of the C axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of U Axis	M3x262	This special M is ON when the servo drive of the U axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of V Axis	M3x263	This special M is ON when the servo drive of the V axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of W Axis	M3x264	This special M is ON when the servo drive of the W axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of 10th Axis	M3x265	This special M is ON when the servo drive of the 10th axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of 11th Axis	M3x266	This special M is ON when the servo drive of the 11th axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of 12th Axis	M3x267	This special M is ON when the servo drive of the 12th axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of 13th Axis	M3x268	This special M is ON when the servo drive of the 13th axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of 14th Axis	M3x269	This special M is ON when the servo drive of the 14th axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Connection Status of 15th Axis	M3x270	This special M is ON when the servo drive of the 15th axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R

B

Function Name	Special M	Description	Device
Servo Connection Status of 16th Axis	M3x271	This special M is ON when the servo drive of the 16th axis is connected. This connection status can be monitored from the [Connect] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of X Axis	M3x272	This special M is ON when the servo drive of the X axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of Y Axis	M3x273	This special M is ON when the servo drive of the Y axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of Z Axis	M3x274	This special M is ON when the servo drive of the Z axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of A Axis	M3x275	This special M is ON when the servo drive of the A axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of B Axis	M3x276	This special M is ON when the servo drive of the B axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of C Axis	M3x277	This special M is ON when the servo drive of the C axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of U Axis	M3x278	This special M is ON when the servo drive of the U axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of V Axis	M3x279	This special M is ON when the servo drive of the V axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of W Axis	M3x280	This special M is ON when the servo drive of the W axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of 10th Axis	M3x281	This special M is ON when the servo drive of the 10th axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of 11th Axis	M3x282	This special M is ON when the servo drive of the 11th axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of 12th Axis	M3x283	This special M is ON when the servo drive of the 12th axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of 13th Axis	M3x284	This special M is ON when the servo drive of the 13th axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of 14th Axis	M3x285	This special M is ON when the servo drive of the 14th axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Servo Enable Status of 15th Axis	M3x286	This special M is ON when the servo drive of the 15th axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R

Function Name	Special M	Description	Device
Servo Enable Status of 16th Axis	M3x287	This special M is ON when the servo drive of the 16th axis is servo ON. This connection status can be monitored from the [Ready] status light of the [Servo Monitor Table] page.	R
Oscillation Control Status of 1st Axis	M3x304	This special M is ON when the 1st axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 2nd Axis	M3x305	This special M is ON when the 2nd axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 3rd Axis	M3x306	This special M is ON when the 3rd axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 4th Axis	M3x307	This special M is ON when the 4th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 5th Axis	M3x308	This special M is ON when the 5th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 6th Axis	M3x309	This special M is ON when the 6th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 7th Axis	M3x310	This special M is ON when the 7th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 8th Axis	M3x311	This special M is ON when the 8th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 9th Axis	M3x312	This special M is ON when the 9th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 10th Axis	M3x313	This special M is ON when the 10th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 11th Axis	M3x314	This special M is ON when the 11th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 12th Axis	M3x315	This special M is ON when the 12th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 13th Axis	M3x316	This special M is ON when the 13th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 14th Axis	M3x317	This special M is ON when the 14th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 15th Axis	M3x318	This special M is ON when the 15th axis has enabled the oscillation function and set movement command.	R
Oscillation Control Status of 16th Axis	M3x319	This special M is ON when the 16th axis has enabled the oscillation function and set movement command.	R
Axis Homed Status of X Axis	M3x320	This special M is ON when the X axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of Y Axis	M3x321	This special M is ON when the Y axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of Z Axis	M3x322	This special M is ON when the Z axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of A Axis	M3x323	This special M is ON when the A axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of B Axis	M3x324	This special M is ON when the B axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of C Axis	M3x325	This special M is ON when the C axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of U Axis	M3x326	This special M is ON when the U axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of V Axis	M3x327	This special M is ON when the V axis is homed and the controller's POS page shows the origin complete symbol.	R

B

Function Name	Special M	Description	Device
Axis Homed Status of W Axis	M3x328	This special M is ON when the W axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of 10th Axis	M3x329	This special M is ON when the 10th axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of 11th Axis	M3x330	This special M is ON when the 11th axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of 12th Axis	M3x331	This special M is ON when the 12th axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of 13th Axis	M3x332	This special M is ON when the 13th axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of 14th Axis	M3x333	This special M is ON when the 14th axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of 15th Axis	M3x334	This special M is ON when the 15th axis is homed, and the controller's POS page shows the origin complete symbol.	R
Axis Homed Status of 16th Axis	M3x335	This special M is ON when the 16th axis is homed, and the controller's POS page shows the origin complete symbol.	R
Homing Finished Status of X Axis	M3x336	This special M is ON when the X axis has finished its homing procedure.	R
Homing Finished Status of Y Axis	M3x337	This special M is ON when the Y axis has finished its homing procedure.	R
Homing Finished Status of Z Axis	M3x338	This special M is ON when the Z axis has finished its homing procedure.	R
Homing Finished Status of A Axis	M3x339	This special M is ON when the A axis has finished its homing procedure.	R
Homing Finished Status of B Axis	M3x340	This special M is ON when the B axis has finished its homing procedure.	R
Homing Finished Status of C Axis	M3x341	This special M is ON when the C axis has finished its homing procedure.	R
Homing Finished Status of U Axis	M3x342	This special M is ON when the U axis has finished its homing procedure.	R
Homing Finished Status of V Axis	M3x343	This special M is ON when the V axis has finished its homing procedure.	R
Homing Finished Status of W Axis	M3x344	This special M is ON when the W axis has finished its homing procedure.	R
Homing Finished Status of 10th Axis	M3x345	This special M is ON when the 10th axis has finished its homing procedure.	R
Homing Finished Status of 11th Axis	M3x346	This special M is ON when the 11th axis has finished its homing procedure.	R
Homing Finished Status of 12th Axis	M3x347	This special M is ON when the 12th axis has finished its homing procedure.	R
Homing Finished Status of 13th Axis	M3x348	This special M is ON when the 13th axis has finished its homing procedure.	R
Homing Finished Status of 14th Axis	M3x349	This special M is ON when the 14th axis has finished its homing procedure.	R
Homing Finished Status of 15th Axis	M3x350	This special M is ON when the 15th axis has finished its homing procedure.	R
Homing Finished Status of 16th Axis	M3x351	This special M is ON when the 16th axis has finished its homing procedure.	R
X Axis at Origin Position	M3x352	This special M is ON when the X axis's machine position is 0.	R
Y Axis at Origin Position	M3x353	This special M is ON when the Y axis's machine position is 0.	R

Function Name	Special M	Description	Device
Z Axis at Origin Position	M3x354	This special M is ON when the Z axis's machine position is 0.	R
A Axis at Origin Position	M3x355	This special M is ON when the A axis's machine position is 0.	R
B Axis at Origin Position	M3x356	This special M is ON when the B axis's machine position is 0.	R
C Axis at Origin Position	M3x357	This special M is ON when the C axis's machine position is 0.	R
U Axis at Origin Position	M3x358	This special M is ON when the U axis's machine position is 0.	R
V Axis at Origin Position	M3x359	This special M is ON when the V axis's machine position is 0.	R
W Axis at Origin Position	M3x360	This special M is ON when the W axis's machine position is 0.	R
10th Axis at Origin Position	M3x361	This special M is ON when the 10th axis's machine position is 0.	R
11th Axis at Origin Position	M3x362	This special M is ON when the 11th axis's machine position is 0.	R
12th Axis at Origin Position	M3x363	This special M is ON when the 12th axis's machine position is 0.	R
13th Axis at Origin Position	M3x364	This special M is ON when the 13th axis's machine position is 0.	R
14th Axis at Origin Position	M3x365	This special M is ON when the 14th axis's machine position is 0.	R
15th Axis at Origin Position	M3x366	This special M is ON when the 15th axis's machine position is 0.	R
16th Axis at Origin Position	M3x367	This special M is ON when the 16th axis's machine position is 0.	R
X Axis Switch to MLC Axis Finished	M3x432	This special M is ON when **M2x432** is triggered, and the X axis has switched to MLC control mode.	R
Y Axis Switch to MLC Axis Finished	M3x433	This special M is ON when **M2x433** is triggered, and the Y axis has switched to MLC control mode.	R
Z Axis Switch to MLC Axis Finished	M3x434	This special M is ON when **M2x434** is triggered, and the Z axis has switched to MLC control mode.	R
A Axis Switch to MLC Axis Finished	M3x435	This special M is ON when **M2x435** is triggered, and the A axis has switched to MLC control mode.	R
B Axis Switch to MLC Axis Finished	M3x436	This special M is ON when **M2x436** is triggered, and the B axis has switched to MLC control mode.	R
C Axis Switch to MLC Axis Finished	M3x437	This special M is ON when **M2x437** is triggered, and the C axis has switched to MLC control mode.	R
U Axis Switch to MLC Axis Finished	M3x438	This special M is ON when **M2x438** is triggered, and the U axis has switched to MLC control mode.	R
V Axis Switch to MLC Axis Finished	M3x439	This special M is ON when **M2x439** is triggered, and the V axis has switched to MLC control mode.	R
W Axis Switch to MLC Axis Finished	M3x440	This special M is ON when **M2x440** is triggered, and the W axis has switched to MLC control mode.	R
10th Axis Switch to MLC Axis Finished	M3x441	This special M is ON when **M2x441** is triggered, and the 10th axis has switched to MLC control mode.	R
11th Axis Switch to MLC Axis Finished	M3x442	This special M is ON when **M2x442** is triggered, and the 11th axis has switched to MLC control mode.	R
12th Axis Switch to MLC Axis Finished	M3x443	This special M is ON when **M2x443** is triggered, and the 12th axis has switched to MLC control mode.	R
13th Axis Switch to MLC Axis Finished	M3x444	This special M is ON when **M2x444** is triggered, and the 13th axis has switched to MLC control mode.	R
14th Axis Switch to MLC Axis Finished	M3x445	This special M is ON when **M2x445** is triggered, and the 14th axis has switched to MLC control mode.	R

B

Function Name	Special M	Description	Device
15th Axis Switch to MLC Axis Finished	M3x446	This special M is ON when **M2x446** is triggered, and the 15th axis has switched to MLC control mode.	R
16th Axis Switch to MLC Axis Finished	M3x447	This special M is ON when **M2x447** is triggered, and the 16th axis has switched to MLC control mode.	R
X Axis Target Reached (MLC Axis)	M3x448	This special M is ON when the X axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x448** will be written to the special M.	R
Y Axis Target Reached (MLC Axis)	M3x449	This special M is ON when the Y axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x449** will be written to the special M.	R
Z Axis Target Reached (MLC Axis)	M3x450	This special M is ON when the Z axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x450** will be written to the special M.	R
A Axis Target Reached (MLC Axis)	M3x451	This special M is ON when the A axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x451** will be written to the special M.	R
B Axis Target Reached (MLC Axis)	M3x452	This special M is ON when the B axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x452** will be written to the special M.	R
C Axis Target Reached (MLC Axis)	M3x453	This special M is ON when the C axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x453** will be written to the special M.	R
U Axis Target Reached (MLC Axis)	M3x454	This special M is ON when the U axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x454** will be written to the special M.	R
V Axis Target Reached (MLC Axis)	M3x455	This special M is ON when the V axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x455** will be written to the special M.	R
W Axis Target Reached (MLC Axis)	M3x456	This special M is ON when the W axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x456** will be written to the special M.	R
10th Axis Target Reached (MLC Axis)	M3x457	This special M is ON when the 10th axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x457** will be written to the special M.	R
11th Axis Target Reached (MLC Axis)	M3x458	This special M is ON when the 11th axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x458** will be written to the special M.	R
12th Axis Target Reached (MLC Axis)	M3x459	This special M is ON when the 12th axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x459** will be written to the special M.	R
13th Axis Target Reached (MLC Axis)	M3x460	This special M is ON when the 13th axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x460** will be written to the special M.	R
14th Axis Target Reached (MLC Axis)	M3x461	This special M is ON when the 14th axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x461** will be written to the special M.	R
15th Axis Target Reached (MLC Axis)	M3x462	This special M is ON when the 15th axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x462** will be written to the special M.	R

Function Name	Special M	Description	Device
16th Axis Target Reached (MLC Axis)	M3x463	This special M is ON when the 16th axis reaches the target position. If the MLC is in speed mode, then the speed represented by **M3x463** will be written to the special M.	R
X Axis Is Moving	M3x464	This special M is ON when the X axis is in motion, regardless of the type of mode.	R
Y Axis Is Moving	M3x465	This special M is ON when the Y axis is in motion, regardless of the type of mode.	R
Z Axis Is Moving	M3x466	This special M is ON when the Z axis is in motion, regardless of the type of mode.	R
A Axis Is Moving	M3x467	This special M is ON when the A axis is in motion, regardless of the type of mode.	R
B Axis Is Moving	M3x468	This special M is ON when the B axis is in motion, regardless of the type of mode.	R
C Axis Is Moving	M3x469	This special M is ON when the C axis is in motion, regardless of the type of mode.	R
U Axis Is Moving	M3x470	This special M is ON when the U axis is in motion, regardless of the type of mode.	R
V Axis Is Moving	M3x471	This special M is ON when the V axis is in motion, regardless of the type of mode.	R
W Axis Is Moving	M3x472	This special M is ON when the W axis is in motion, regardless of the type of mode.	R
10th Axis Is Moving	M3x473	This special M is ON when the 10th axis is in motion, regardless of the type of mode.	R
11th Axis Is Moving	M3x474	This special M is ON when the 11th axis is in motion, regardless of the type of mode.	R
12th Axis Is Moving	M3x475	This special M is ON when the 12th axis is in motion, regardless of the type of mode.	R
13th Axis Is Moving	M3x476	This special M is ON when the 13th axis is in motion, regardless of the type of mode.	R
14th Axis Is Moving	M3x477	This special M is ON when the 14th axis is in motion, regardless of the type of mode.	R
15th Axis Is Moving	M3x478	This special M is ON when the 15th axis is in motion, regardless of the type of mode.	R
16th Axis Is Moving	M3x479	This special M is ON when the 16th axis is in motion, regardless of the type of mode.	R
X Axis Moving Forward	M3x480	This special M is ON when the X axis is moving in the positive direction.	R
Y Axis Moving Forward	M3x481	This special M is ON when the Y axis is moving in the positive direction.	R
Z Axis Moving Forward	M3x482	This special M is ON when the Z axis is moving in the positive direction.	R
A Axis Moving Forward	M3x483	This special M is ON when the A axis is moving in the positive direction.	R
B Axis Moving Forward	M3x484	This special M is ON when the B axis is moving in the positive direction.	R
C Axis Moving Forward	M3x485	This special M is ON when the C axis is moving in the positive direction.	R
U Axis Moving Forward	M3x486	This special M is ON when the U axis is moving in the positive direction.	R
V Axis Moving Forward	M3x487	This special M is ON when the V axis is moving in the positive direction.	R
W Axis Moving Forward	M3x488	This special M is ON when the W axis is moving in the positive direction.	R
10th Axis Moving Forward	M3x489	This special M is ON when the 10th axis is moving in the positive direction.	R
11th Axis Moving Forward	M3x490	This special M is ON when the 11th axis is moving in the positive direction.	R
12th Axis Moving Forward	M3x491	This special M is ON when the 12th axis is moving in the positive direction.	R
13th Axis Moving Forward	M3x492	This special M is ON when the 13th axis is moving in the positive direction.	R

B

Function Name	Special M	Description	Device
14th Axis Moving Forward	M3x493	This special M is ON when the 14th axis is moving in the positive direction.	R
15th Axis Moving Forward	M3x494	This special M is ON when the 15th axis is moving in the positive direction.	R
16th Axis Moving Forward	M3x495	This special M is ON when the 16th axis is moving in the positive direction.	R
X Axis Moving Backward	M3x496	This special M is ON when the X axis is moving in the negative direction.	R
Y Axis Moving Backward	M3x497	This special M is ON when the Y axis is moving in the negative direction.	R
Z Axis Moving Backward	M3x498	This special M is ON when the Z axis is moving in the negative direction.	R
A Axis Moving Backward	M3x499	This special M is ON when the A axis is moving in the negative direction.	R
B Axis Moving Backward	M3x500	This special M is ON when the B axis is moving in the negative direction.	R
C Axis Moving Backward	M3x501	This special M is ON when the C axis is moving in the negative direction.	R
U Axis Moving Backward	M3x502	This special M is ON when the U axis is moving in the negative direction.	R
V Axis Moving Backward	M3x503	This special M is ON when the V axis is moving in the negative direction.	R
W Axis Moving Backward	M3x504	This special M is ON when the W axis is moving in the negative direction.	R
10th Axis Moving Backward	M3x505	This special M is ON when the 10th axis is moving in the negative direction.	R
11th Axis Moving Backward	M3x506	This special M is ON when the 11th axis is moving in the negative direction.	R
12th Axis Moving Backward	M3x507	This special M is ON when the 12th axis is moving in the negative direction.	R
13th Axis Moving Backward	M3x508	This special M is ON when the 13th axis is moving in the negative direction.	R
14th Axis Moving Backward	M3x509	This special M is ON when the 14th axis is moving in the negative direction.	R
15th Axis Moving Backward	M3x510	This special M is ON when the 15th axis is moving in the negative direction.	R
16th Axis Moving Backward	M3x511	This special M is ON when the 16th axis is moving in the negative direction.	R
Diameter or Radius Mode of X Axis	M3x512	In the lathe system, this special M is ON when the X axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of Y Axis	M3x513	In the lathe system, this special M is ON when the Y axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of Z Axis	M3x514	In the lathe system, this special M is ON when the Z axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of A Axis	M3x515	In the lathe system, this special M is ON when the A axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of B Axis	M3x516	In the lathe system, this special M is ON when the B axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of C Axis	M3x517	In the lathe system, this special M is ON when the C axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of U Axis	M3x518	In the lathe system, this special M is ON when the U axis is in the diameter mode or OFF when it is in the radius mode.	R

Function Name	Special M	Description	Device
Diameter or Radius Mode of V Axis	M3x519	In the lathe system, this special M is ON when the V axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of W Axis	M3x520	In the lathe system, this special M is ON when the W axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of 10th Axis	M3x521	In the lathe system, this special M is ON when the 10th axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of 11th Axis	M3x522	In the lathe system, this special M is ON when the 11th axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of 12th Axis	M3x523	In the lathe system, this special M is ON when the 12th axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of 13th Axis	M3x524	In the lathe system, this special M is ON when the 13th axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of 14th Axis	M3x525	In the lathe system, this special M is ON when the 14th axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of 15th Axis	M3x526	In the lathe system, this special M is ON when the 15th axis is in the diameter mode or OFF when it is in the radius mode.	R
Diameter or Radius Mode of 16th Axis	M3x527	In the lathe system, this special M is ON when the 16th axis is in the diameter mode or OFF when it is in the radius mode.	R
1st Spindle Speed Reach	M3x704	This special M is ON when the 1st spindle's speed reaches the target value.	R
1st Spindle Zero Speed	M3x705	This special M is ON when the 1st spindle's speed reaches zero.	R
1st Spindle Target Reach	M3x706	This special M is ON when the 1st spindle reaches the target position.	R
1st Spindle Is in The Rigid Tapping	M3x707	This special M is ON when the 1st spindle is executing the rigid tapping.	R
1st Spindle Is in Position Axis Mode	M3x709	This special M is ON when the 1st spindle is switching from S axis to C axis.	R
1st Spindle Ready	M3x710	This special M is ON when the 1st spindle is ready to use.	R
2nd Spindle Speed Reach	M3x720	This special M is ON when the 2nd spindle's speed reaches the target value.	R
2nd Spindle Zero Speed	M3x721	This special M is ON when the 2nd spindle's speed reaches zero.	R
2nd Spindle Target Reach	M3x722	This special M is ON when the 2nd spindle reaches the target position.	R
2nd Spindle Is in The Rigid Tapping	M3x723	This special M is ON when the 2nd spindle is executing the rigid tapping.	R
2nd Spindle Is in Position Axis Mode	M3x725	This special M is ON when the 2nd spindle is switching from S axis to C axis.	R
2nd Spindle Ready	M3x726	This special M is ON when the 2nd spindle is ready to use.	R
3rd Spindle Speed Reach	M3x736	This special M is ON when the 3rd spindle's speed reaches the target value.	R
3rd Spindle Zero Speed	M3x737	This special M is ON when the 3rd spindle's speed reaches zero.	R
3rd Spindle Target Reach	M3x738	This special M is ON when the 3rd spindle reaches the target position.	R
3rd Spindle Is in The Rigid Tapping	M3x739	This special M is ON when the 3rd spindle is executing the rigid tapping.	R
3rd Spindle Is in Position Axis Mode	M3x741	This special M is ON when the 3rd spindle is switching from S axis to C axis.	R

B

Function Name	Special M	Description	Device
3rd Spindle Ready	M3x742	This special M is ON when the 3rd spindle is ready to use.	R
4th Spindle Speed Reach	M3x752	This special M is ON when the 4th spindle's speed reaches the target value.	R
4th Spindle Zero Speed	M3x753	This special M is ON when the 4th spindle's speed reaches zero.	R
4th Spindle Target Reach	M3x754	This special M is ON when the 4th spindle reaches the target position.	R
4th Spindle Is in The Rigid Tapping	M3x755	This special M is ON when the 4th spindle is executing the rigid tapping.	R
4th Spindle Is in Position Axis Mode	M3x757	This special M is ON when the 4th spindle is switching from S axis to C axis.	R
4th Spindle Ready	M3x758	This special M is ON when the 4th spindle is ready to use.	R
5th Spindle Speed Reach	M3x768	This special M is ON when the 5th spindle's speed reaches the target value.	R
5th Spindle Zero Speed	M3x769	This special M is ON when the 5th spindle's speed reaches zero.	R
5th Spindle Positioning Control	M3x770	This special M is ON when the 5th spindle reaches the target position.	R
5th Spindle Is in The Rigid Tapping	M3x771	This special M is ON when the 5th spindle is executing the rigid tapping.	R
Switching C / S Axis of 5th Lathe Spindle	M3x773	This special M is ON when the 5th spindle is switching from S axis to C axis.	R
5th Spindle Ready	M3x774	This special M is ON when the 5th spindle is ready to use.	R
6th Spindle Speed Reach	M3x784	This special M is ON when the 6th spindle's speed reaches the target value.	R
6th Spindle Zero Speed	M3x785	This special M is ON when the 6th spindle's speed reaches zero.	R
6th Spindle Target Reach	M3x786	This special M is ON when the 6th spindle reaches the target position.	R
6th Spindle Is in The Rigid Tapping	M3x787	This special M is ON when the 6th spindle is executing the rigid tapping.	R
6th Spindle Is in Position Axis Mode	M3x789	This special M is ON when the 6th spindle is switching from S axis to C axis.	R
6th Spindle Ready	M3x790	This special M is ON when the 6th spindle is ready to use.	R
7th Spindle Speed Reach	M3x800	This special M is ON when the 7th spindle's speed reaches the target value.	R
7th Spindle Zero Speed	M3x801	This special M is ON when the 7th spindle's speed reaches zero.	R
7th Spindle Target Reach	M3x802	This special M is ON when the 7th spindle reaches the target position.	R
7th Spindle Is in The Rigid Tapping	M3x803	This special M is ON when the 7th spindle is executing the rigid tapping.	R
7th Spindle Is in Position Axis Mode	M3x805	This special M is ON when the 7th spindle is switching from S axis to C axis.	R
7th Spindle Ready	M3x806	This special M is ON when the 7th spindle is ready to use.	R
8th Spindle Speed Reach	M3x816	This special M is ON when the 8th spindle's speed reaches the target value.	R
8th Spindle Zero Speed	M3x817	This special M is ON when the 8th spindle's speed reaches zero.	R
8th Spindle Target Reach	M3x818	This special M is ON when the 8th spindle reaches the target position.	R
8th Spindle Is in The Rigid Tapping	M3x819	This special M is ON when the 8th spindle is executing the rigid tapping.	R

Function Name	Special M	Description	Device
8th Spindle Is in Position Axis Mode	M3x821	This special M is ON when the 8th spindle is switching from S axis to C axis.	R
8th Spindle Ready	M3x822	This special M is ON when the 8th spindle is ready to use.	R

B.2.3 List of special D (MLC to system)

B.2.3.1 D20000 to D28999

B

Function Name	Special D	Description	Device	Type	Range
NC Channel Selection	D20000	Setting the current NC channel for the controller and screen interface. When the value is 1, it means the system is using and showing NC channel 1; when the value is 2, it means the system is using and showing NC channel 2, and so forth.	R/W	Decimal	0 ~ 65,535
Spindle Analog Voltage Output Port 1	D20160	When the spindle is in EtherCAT mode, users can use this special D register to output the analog voltage. Unit: 0.01V.	R/W	Decimal	-1,000 ~ +1,000
Spindle Analog Voltage Output Port 2	D20161		R/W	Decimal	-1,000 ~ +1,000
1st Remote Module Output	D20200 D20201 D20202 D20203	For connected modules that do not include digital input/output (DI/O), the system will set the arrangement order according to the EtherCAT module, and will set the special D module with the special D based on the connection order (such as analog modules). After the remote modules get these data, it will output the results as transferred data. Taking the first module as an example, D20200 corresponds to the first set of values on the module, D20201 corresponds to the second set of values on the module, and so forth.	R/W	Decimal	0 ~ 65,535
2nd Remote Module Output	D20204 D20205 D20206 D20207		R/W	Decimal	0 ~ 65,535
3rd Remote Module Output	D20208 D20209 D20210 D20211		R/W	Decimal	0 ~ 65,535
4th Remote Module Output	D20212 D20213 D20214 D20215		R/W	Decimal	0 ~ 65,535
5th Remote Module Output	D20216 D20217 D20218 D20219		R/W	Decimal	0 ~ 65,535
6th Remote Module Output	D20220 D20221 D20222 D20223		R/W	Decimal	0 ~ 65,535
7th Remote Module Output	D20224 D20225 D20226 D20227		R/W	Decimal	0 ~ 65,535
8th Remote Module Output	D20228 D20229 D20230 D20231		R/W	Decimal	0 ~ 65,535
NC Mode Switching	D2x000	This special D is for NC channels to switch to different operation modes. 0: AUTO 1: EDIT 2: MDI 3: MPG 4: JOG 5: RAPID 6: INC 7: HOME	R/W	Decimal	0 ~ 7

Function Name	Special D	Description	Device	Type	Range
Feed Rate Percentage	D2x002	Setting for feed rate percentage of NC program speed. Unit: % Ex: When the NC program speed is 1000 mm/min and this D2x002 is 50, this means the NC system will execute the axes interpolation speed as 1000 x 50% = 500 mm/min.	R/W	Decimal	0 ~ 150
Rapid Speed Percentage	D2x004	Setting NC rapid speed percentage for G00 command. Unit: % Ex: When the NC rapid speed [N1.030] is 6000 mm/min and this D2x004 is 50, this means the NC system will execute the axes rapid speed as 6000 x 50% = 3000 mm/min.	R/W	Decimal	0 ~ 65,535
Speed Override for JOG and INC	D2x006	Setting JOG and INC speed. When the system is in JOG or INC mode, it will take two different mode types to determine the moving speed. If the JOG speed mode **[N1.011 Bit26]** is set as 0, the system will take the JOG maximum speed **[N2.030]** and then multiply this the value in this special D (unit: %) as the JOG moving speed. If the JOG speed mode **[N1.011 Bit26]** is set as 1, the system will take the value of this special D as the JOG moving speed directly. The unit of this special D will refer to parameter axes control type **[N2.001]**. N2.001 / Unit Bit 2~4 (Linear axis) / mm/min Bit 2~4 (Rotary axis) Bit 11 = 0 / Deg/min Bit 2~4 (Rotary axis) Bit 11 = 1 / RPM	R/W	Decimal	Percentage Mode 0 ~ 100 Constant Mode 0 ~ 65,535
1st MPG Axes Selection	D2x008	This special D is for the NC system to switch the 1st MPG axis in MPG mode. 0: X axis 1: Y axis 2: Z axis 3: A axis … 15: 16th axis	R/W	Decimal	0 ~ 15
1st MPG Ratio Selection	D2x009	This special D is for the NC system to switch the 1st MPG axis moving ratio, which can be 1, 10 or 100. The system will take the unit setting **[N9.013]** as the smallest movement and then multiply it by this ratio to derive the final movement on each scale of the MPG hand wheel. Ex: When D2x009 is 1 and the unit setting [N9.013] is 0.001, with 3 decimal places, the minimum movement of the MPG hand wheel will be 0.001 × 1 = 0.001 mm.	R/W	Decimal	0 ~ 65,535

B

Function Name	Special D	Description	Device	Type	Range
2nd MPG Axes Selection	D2x010	This special D is for the NC system to switch the 2nd MPG axis in MPG mode. 0: X axis 1: Y axis 2: Z axis 3: A axis ... 15: 16th axis *Set M20024 **[Enable 3 MPG Control]** to ON to enable this function.	R/W	Decimal	0 ~ 15
2nd MPG Ratio Selection	D2x011	This special D is for the NC system to switch the 2nd MPG axis moving ratio, which can be 1, 10 or 100. The system will take the unit setting **[N9.013]** as the smallest movement and then multiply it by this ratio to derive the final movement on each scale of the MPG hand wheel. Ex: When D2x009 is 1 and the unit setting **[N9.013]** is 0.001, with 3 decimal places, the minimum movement of the MPG hand wheel will be 0.001 × 1 = 0.001 mm. *Set M20024 **[Enable 3 MPG Control]** to ON to enable this function.	R/W	Decimal	0 ~ 65,535
3rd MPG Axes Selection	D2x012	This special D is for the NC system to switch the MPG axis in the 3rd MPG mode. 0: X axis 1: Y axis 2: Z axis 3: A axis ... 15: 16th axis *Set M20024 **[Enable 3 MPG Control]** to ON to enable this function.	R/W	Decimal	0 ~ 15
3rd MPG Ratio Selection	D2x013	This special D is for the NC system to switch the 3rd MPG axis moving ratio, which can be 1, 10 or 100. The system will take the unit setting **[N9.013]** as the smallest movement and then multiply it by this ratio to derive the final movement on each scale of the MPG hand wheel. Ex: When D2x009 is 1 and the unit setting **[N9.013]** is 0.001, with 3 decimal places, the minimum movement of the MPG hand wheel will be 0.001 × 1 = 0.001 mm. *Set M20024 **[Enable 3 MPG Control]** to ON to enable this function.	R/W	Decimal	0 ~ 65,535

Function Name	Special D	Description	Device	Type	Range
Axes Movement in INC Mode	D2x014	When the NC system is in INC mode and the JOG motion trigger special M (M2x384 ~ M2x415) is enabled, the system will take this special D and then multiply the unit setting **[N9.013]** to derive the target movement. Ex: When D2x014 is 1234 and unit setting [N9.013] is 0.001, with 3 decimal places, the axes will move 1.234 mm every time the special M of JOG motion trigger enabled.	R/W	Decimal	0 ~ 4,294,967,295
Robot's Coordinate System Switch in Manual Mode	D2x016	Setting the Robot's coordinate system when the system is in manual mode. 0: Not using coordinate manual mode 1 ~ 6: corresponding to G54 ~ G59.	R/W	Decimal	0 ~ 6
Robot's Tool System Switch in Manual Mode	D2x017	Setting the Robot's tool offset system when the system is in manual mode. 0: Not using tool offset 1 ~ N: corresponds to tool number	R/W	Decimal	0 ~ N N will be based on N0.408 ~ N0.411
Axes Oscillation Amplitude	D2x018 D2x019	When the axes oscillation control **[M2x264]** is enabled, the activated axes will oscillate based on the amplitude here. Unit: mm	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Axes Oscillation Cycle Time	D2x020 D2x021	When the axes oscillation control **[M2x264]** is enabled, the system will take this special D as the cycle time for each oscillation command. Unit: ms	R/W	Float	2,147,483,648 ~ +2,147,483,647
Oscillation Axes Enable	D2x022	The oscillation axes use this 16-bit special D as the mask to determine whether to enable the function on each specific axis. For example, if this set to 5, it means the X and Z axes will both have their oscillation function activated.	R/W	Decimal	0 ~ 65,535
Axes Oscillation Wave Type	D2x023	Setting for axes oscillation wave type. 0: Start position as middle position of the SIN wave. 1: Start position as base position (max or min) of the SIN wave.	R/W	Decimal	0 ~ 1
1st Spindle Speed	D2x024 D2x025	Write the 1st spindle's speed through the special D (in accordance with **M2x710**).	R/W	Decimal	0 ~ 4,294,967,295
1st Spindle Speed Rate	D2x026	Setting the spindle's speed ratio. Ex: When the program speed is S1000 and this special D is 30, then the NC system will execute the spindle speed as 1000 x 30% = 300 RPM.	R/W	Decimal	0 ~ 65,535
1st Spindle Gear Ratio Selection	D2x027	Select in accordance with **N0.1034 ~ N0.1041**. Ex: When this special D is set to 1, the system will set the 1st spindle to 1st gear, when it is set to 2, it will be in 2nd gear, and so forth.	R/W	Decimal	0 ~ 65,535

B

Function Name	Special D	Description	Device	Type	Range
2nd Spindle Speed	D2x030 D2x031	Write the 2nd spindle's speed through the special D (in accordance with **M2x726**).	R/W	Decimal	0 ~ 4,294,967,295
2nd Spindle Speed Rate	D2x032	Setting the spindle's speed ratio. Ex: When the program speed is S1000 and this special D is 30, then the NC system will execute the spindle speed as 1000 x 30% = 300 RPM.	R/W	Decimal	0 ~ 65,535
2nd Spindle Gear Ratio Selection	D2x033	Select in accordance with **N0.1084 ~ N0.1091**. Ex: When this special D is set to 1, the system will set the 2nd spindle to 1st gear, when it is set to 2, it will be in 2nd gear, and so forth.	R/W	Decimal	0 ~ 65,535
Tool Number Tool Magazine 1	D2x036	Write the tool number through the special D (in accordance with **N1.010**).	R/W	Decimal	0 ~ 65,535
Standby Tool Number Tool Magazine 1	D2x037	Write the standby tool number through the special D (in accordance with **N1.010**).	R/W	Decimal	0 ~ 65,535
Command Tool Number Tool Magazine 1	D2x038	Write the command tool number through the special D (in accordance with **N1.010**).	R/W	Decimal	0 ~ 65,535
Tool Number Tool Magazine 2	D2x042	Write the tool number through the special D (in accordance with **N1.010**).	R/W	Decimal	0 ~ 65,535
Standby Tool Number Tool Magazine 2	D2x043	Write the standby tool number through the special D (in accordance with **N1.010**).	R/W	Decimal	0 ~ 65,535
Command Tool Number Tool Magazine 2	D2x044	Write the command tool number through the special D (in accordance with **N1.010**).	R/W	Decimal	0 ~ 65,535
1st Macro Call Macro Number	D2x064	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 1st macro call is triggered with special **M2x032**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
2nd Macro Call Macro Number	D2x065	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 2nd macro call is triggered with special **M2x033**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
3rd Macro Call Macro Number	D2x066	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 3rd macro call is triggered with special **M2x034**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535

Function Name	Special D	Description	Device	Type	Range
4th Macro Call Macro Number	D2x067	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 4th macro call is triggered with special **M2x035**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
5th Macro Call Macro Number	D2x068	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 5th macro call is triggered with special **M2x036**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
6th Macro Call Macro Number	D2x069	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 6th macro call is triggered with special **M2x037**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
7th Macro Call Macro Number	D2x070	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 7th macro call is triggered with special **M2x038**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
8th Macro Call Macro Number	D2x071	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 8th macro call is triggered with special **M2x039**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
9th Macro Call Macro Number	D2x072	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 9th macro call is triggered with special **M2x040**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
10th Macro Call Macro Number	D2x073	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 10th macro call is triggered with special **M2x041**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
11th Macro Call Macro Number	D2x074	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 11th macro call is triggered with special **M2x042**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
12th Macro Call Macro Number	D2x075	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 12th macro call is triggered with special **M2x043**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535

B

Function Name	Special D	Description	Device	Type	Range
13th Macro Call Macro Number	D2x076	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 13th macro call is triggered with special **M2x044**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
14th Macro Call Macro Number	D2x077	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 14th macro call is triggered with special **M2x045**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
15th Macro Call Macro Number	D2x078	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 15th macro call is triggered with special **M2x046**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
16th Macro Call Macro Number	D2x079	Specify the macro call function's O macro file number, such as O9xxx. Ex: When this special D is written to K9100 and the 16th macro call is triggered with special **M2x047**, the NC system will execute the O9100 program.	R/W	Decimal	1 ~ 65,535
MLC to NC Variable 1	D2x128	The system will move data from this special D to NC variable #25128.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 2	D2x129	The system will move data from this special D to NC variable #25129. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 3	D2x130	The system will move data from this special D to NC variable #25130.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 4	D2x131	The system will move data from this special D to NC variable #25131. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 5	D2x132	The system will move data from this special D to NC variable #25132.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 6	D2x133	The system will move data from this special D to NC variable #25133. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 7	D2x134	The system will move data from this special D to NC variable #25134.	R/W	Decimal	0 ~ 65,535

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 8	D2x135	The system will move data from this special D to NC variable #25135. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 9	D2x136	The system will move data from this special D to NC variable #25136.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 10	D2x137	The system will move data from this special D to NC variable #25137. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 11	D2x138	The system will move data from this special D to NC variable #25138.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 12	D2x139	The system will move data from this special D to NC variable #25139. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 13	D2x140	The system will move data from this special D to NC variable #25140.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 14	D2x141	The system will move data from this special D to NC variable #25141. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 15	D2x142	The system will move data from this special D to NC variable #25142.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 16	D2x143	The system will move data from this special D to NC variable #25143. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 17	D2x144	The system will move data from this special D to NC variable #25144.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 18	D2x145	The system will move data from this special D to NC variable #25145. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535

B

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 19	D2x146	The system will move data from this special D to NC variable #25146.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 20	D2x147	The system will move data from this special D to NC variable #25147. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 21	D2x148	The system will move data from this special D to NC variable #25148.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 22	D2x149	The system will move data from this special D to NC variable #25149. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 23	D2x150	The system will move data from this special D to NC variable #25150.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 24	D2x151	The system will move data from this special D to NC variable #25151. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 25	D2x152	The system will move data from this special D to NC variable #25152.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 26	D2x153	The system will move data from this special D to NC variable #25153. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 27	D2x154	The system will move data from this special D to NC variable #25154.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 28	D2x155	The system will move data from this special D to NC variable #25155. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 29	D2x156	The system will move data from this special D to NC variable #25156.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 30	D2x157	The system will move data from this special D to NC variable #25157. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 31	D2x158	The system will move data from this special D to NC variable #25158.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 32	D2x159	The system will move data from this special D to NC variable #25159. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 33	D2x160	The system will move data from this special D to NC variable #25160.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 34	D2x161	The system will move data from this special D to NC variable #25161. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 35	D2x162	The system will move data from this special D to NC variable #25162.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 36	D2x163	The system will move data from this special D to NC variable #25163. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 37	D2x164	The system will move data from this special D to NC variable #25164.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 38	D2x165	The system will move data from this special D to NC variable #25165. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 39	D2x166	The system will move data from this special D to NC variable #25166.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 40	D2x167	The system will move data from this special D to NC variable #25167. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 41	D2x168	The system will move data from this special D to NC variable #25168.	R/W	Decimal	0 ~ 65,535

B

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 42	D2x169	The system will move data from this special D to NC variable #25169. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 43	D2x170	The system will move data from this special D to NC variable #25170.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 44	D2x171	The system will move data from this special D to NC variable #25171. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 45	D2x172	The system will move data from this special D to NC variable #25172.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 46	D2x173	The system will move data from this special D to NC variable #25173. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 47	D2x174	The system will move data from this special D to NC variable #25174.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 48	D2x175	The system will move data from this special D to NC variable #25175. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 49	D2x176	The system will move data from this special D to NC variable #25176.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 50	D2x177	The system will move data from this special D to NC variable #25177. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 51	D2x178	The system will move data from this special D to NC variable #25178.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 52	D2x179	The system will move data from this special D to NC variable #25179. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 53	D2x180	The system will move data from this special D to NC variable #25180.	R/W	Decimal	0 ~ 65,535

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 54	D2x181	The system will move data from this special D to NC variable #25181. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 55	D2x182	The system will move data from this special D to NC variable #25182.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 56	D2x183	The system will move data from this special D to NC variable #25183. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 57	D2x184	The system will move data from this special D to NC variable #25184.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 58	D2x185	The system will move data from this special D to NC variable #25185. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 59	D2x186	The system will move data from this special D to NC variable #25186.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 60	D2x187	The system will move data from this special D to NC variable #25187. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 61	D2x188	The system will move data from this special D to NC variable #25188.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 62	D2x189	The system will move data from this special D to NC variable #25189. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 63	D2x190	The system will move data from this special D to NC variable #25190.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 64	D2x191	The system will move data from this special D to NC variable #25191. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535

B

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 65	D2x192	The system will move data from this special D to NC variable #25192.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 66	D2x193	The system will move data from this special D to NC variable #25193. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 67	D2x194	The system will move data from this special D to NC variable #25194.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 68	D2x195	The system will move data from this special D to NC variable #25195. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 69	D2x196	The system will move data from this special D to NC variable #25196.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 70	D2x197	The system will move data from this special D to NC variable #25197. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 71	D2x198	The system will move data from this special D to NC variable #25198.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 72	D2x199	The system will move data from this special D to NC variable #25199. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 73	D2x200	The system will move data from this special D to NC variable #25200.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 74	D2x201	The system will move data from this special D to NC variable #25201. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 75	D2x202	The system will move data from this special D to NC variable #25202.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 76	D2x203	The system will move data from this special D to NC variable #25203. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 77	D2x204	The system will move data from this special D to NC variable #25204.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 78	D2x205	The system will move data from this special D to NC variable #25205. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 79	D2x206	The system will move data from this special D to NC variable #25206.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 80	D2x207	The system will move data from this special D to NC variable #25207. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 81	D2x208	The system will move data from this special D to NC variable #25208.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 82	D2x209	The system will move data from this special D to NC variable #25209. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 83	D2x210	The system will move data from this special D to NC variable #25210.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 84	D2x211	The system will move data from this special D to NC variable #25211. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 85	D2x212	The system will move data from this special D to NC variable #25212.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 86	D2x213	The system will move data from this special D to NC variable #25213. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 87	D2x214	The system will move data from this special D to NC variable #25214.	R/W	Decimal	0 ~ 65,535

B

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 88	D2x215	The system will move data from this special D to NC variable #25215. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 89	D2x216	The system will move data from this special D to NC variable #25216.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 90	D2x217	The system will move data from this special D to NC variable #25217. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 91	D2x218	The system will move data from this special D to NC variable #25218.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 92	D2x219	The system will move data from this special D to NC variable #25219. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 93	D2x220	The system will move data from this special D to NC variable #25220.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 94	D2x221	The system will move data from this special D to NC variable #25221. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 95	D2x222	The system will move data from this special D to NC variable #25222.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 96	D2x223	The system will move data from this special D to NC variable #25223. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 97	D2x224	The system will move data from this special D to NC variable #25224.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 98	D2x225	The system will move data from this special D to NC variable #25225. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 99	D2x226	The system will move data from this special D to NC variable #25226.	R/W	Decimal	0 ~ 65,535

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 100	D2x227	The system will move data from this special D to NC variable #25227. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 101	D2x228	The system will move data from this special D to NC variable #25228.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 102	D2x229	The system will move data from this special D to NC variable #25229. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 103	D2x230	The system will move data from this special D to NC variable #25230.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 104	D2x231	The system will move data from this special D to NC variable #25231. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 105	D2x232	The system will move data from this special D to NC variable #25232.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 106	D2x233	The system will move data from this special D to NC variable #25233. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 107	D2x234	The system will move data from this special D to NC variable #25234.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 108	D2x235	The system will move data from this special D to NC variable #25235. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 109	D2x236	The system will move data from this special D to NC variable #25236.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 110	D2x237	The system will move data from this special D to NC variable #25237. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535

B

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 111	D2x238	The system will move data from this special D to NC variable #25238.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 112	D2x239	The system will move data from this special D to NC variable #25239. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 113	D2x240	The system will move data from this special D to NC variable #25240.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 114	D2x241	The system will move data from this special D to NC variable #25241. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 115	D2x242	The system will move data from this special D to NC variable #25242.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 116	D2x243	The system will move data from this special D to NC variable #25243. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 117	D2x244	The system will move data from this special D to NC variable #25244.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 118	D2x245	The system will move data from this special D to NC variable #25245. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 119	D2x246	The system will move data from this special D to NC variable #25246.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 120	D2x247	The system will move data from this special D to NC variable #25247. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 121	D2x248	The system will move data from this special D to NC variable #25248.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 122	D2x249	The system will move data from this special D to NC variable #25249. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535

Function Name	Special D	Description	Device	Type	Range
MLC to NC Variable 123	D2x250	The system will move data from this special D to NC variable #25250.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 124	D2x251	The system will move data from this special D to NC variable #25251. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 125	D2x252	The system will move data from this special D to NC variable #25252.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 126	D2x253	The system will move data from this special D to NC variable #25253. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 127	D2x254	The system will move data from this special D to NC variable #25254.	R/W	Decimal	0 ~ 65,535
MLC to NC Variable 128	D2x255	The system will move data from this special D to NC variable #25255. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R/W	Decimal	0 ~ 65,535
Target Position of X axis (MLC Axis)	D2x256 D2x257	Specifies the target position of the X axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of Y axis (MLC Axis)	D2x258 D2x259	Specifies the target position of the Y axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of Z axis (MLC Axis)	D2x260 D2x261	Specifies the target position of the Z axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of A axis (MLC Axis)	D2x262 D2x263	Specifies the target position of the A axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of B axis (MLC Axis)	D2x264 D2x265	Specifies the target position of the B axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of C axis (MLC Axis)	D2x266 D2x267	Specifies the target position of the C axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of U axis (MLC Axis)	D2x268 D2x269	Specifies the target position of the U axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647

B

Function Name	Special D	Description	Device	Type	Range
Target Position of V axis (MLC Axis)	D2x270 D2x271	Specifies the target position of the V axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of W axis (MLC Axis)	D2x272 D2x273	Specifies the target position of the W axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of 10th axis (MLC Axis)	D2x274 D2x275	Specifies the target position of the 10th axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of 11th axis (MLC Axis)	D2x276 D2x277	Specifies the target position of the 11th axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of 12th axis (MLC Axis)	D2x278 D2x279	Specifies the target position of the 12th axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of 13th axis (MLC Axis)	D2x280 D2x281	Specifies the target position of the 13th axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of 14th axis (MLC Axis)	D2x282 D2x283	Specifies the target position of the 14th axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of 15th axis (MLC Axis)	D2x284 D2x285	Specifies the target position of the 15th axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Position of 16th axis (MLC Axis)	D2x286 D2x287	Specifies the target position of the 16th axis in MLC axis mode. Unit: mm or inch.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of X axis (MLC Axis)	D2x288 D2x289	Specifies the target velocity of the X axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of Y axis (MLC Axis)	D2x290 D2x291	Specifies the target velocity of the Y axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of Z axis (MLC Axis)	D2x292 D2x293	Specifies the target velocity of the Z axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of A axis (MLC Axis)	D2x294 D2x295	Specifies the target velocity of the A axis in MLC axis mode. Unit: RPM.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of B axis (MLC Axis)	D2x296 D2x297	Specifies the target velocity of the B axis in MLC axis mode. Unit: RPM.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of C axis (MLC Axis)	D2x298 D2x299	Specifies the target velocity of the C axis in MLC axis mode. Unit: RPM.	R/W	Float	-2,147,483,648 ~ +2,147,483,647

Function Name	Special D	Description	Device	Type	Range
Target Velocity of U axis (MLC Axis)	D2x300 D2x301	Specifies the target velocity of the U axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of V axis (MLC Axis)	D2x302 D2x303	Specifies the target velocity of the V axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of W axis (MLC Axis)	D2x304 D2x305	Specifies the target velocity of the W axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of 10th axis (MLC Axis)	D2x306 D2x307	Specifies the target velocity of the 10th axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of 11th axis (MLC Axis)	D2x308 D2x309	Specifies the target velocity of the 11th axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of 12th axis (MLC Axis)	D2x310 D2x311	Specifies the target velocity of the 12th axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of 13th axis (MLC Axis)	D2x312 D2x313	Specifies the target velocity of the 13th axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of 14th axis (MLC Axis)	D2x314 D2x315	Specifies the target velocity of the 14th axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of 15th axis (MLC Axis)	D2x316 D2x317	Specifies the target velocity of the 15th axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
Target Velocity of 16th axis (MLC Axis)	D2x318 D2x319	Specifies the target velocity of the 16th axis in MLC axis mode. Unit: mm/min, inch/min.	R/W	Float	-2,147,483,648 ~ +2,147,483,647
3rd Spindle Speed	D2x320 D2x321	Write the 3rd spindle's speed through the special D (in accordance with **M2x742**).	R/W	Decimal	0 ~ 4,294,967,295
3rd Spindle Speed Rate	D2x322	Setting the spindle's speed ratio. Ex: When the program speed is S1000 and this special D is 30, then the NC system will execute the spindle speed as 1000 x 30% = 300 RPM.	R/W	Decimal	0 ~ 65,535
3rd Spindle Gear Ratio Selection	D2x323	Select in accordance with **N0.1134 ~ N0.1141**. Ex: When this special D is set to 1, the system will set the 3rd spindle to 1st gear, when it is set to 2, it will be in 2nd gear, and so forth.	R/W	Decimal	0 ~ 65,535
4th Spindle Speed	D2x326 D2x327	Write the 4th spindle's speed through the special D (in accordance with **M2x758**).	R/W	Decimal	0 ~ 4,294,967,295

B

Function Name	Special D	Description	Device	Type	Range
4th Spindle Speed Rate	D2x328	Setting the spindle's speed ratio. Ex: When the program speed is S1000 and this special D is 30, then the NC system will execute the spindle speed as 1000 x 30% = 300 RPM.	R/W	Decimal	0 ~ 65,535
4th Spindle Gear Ratio Selection	D2x329	Select in accordance with **N0.1184 ~ N0.1191**. Ex: When this special D is set to 1, the system will set the 4th spindle to 1st gear, when it is set to 2, it will be in 2nd gear, and so forth.	R/W	Decimal	0 ~ 65,535
5th Spindle Speed	D2x332 D2x333	Write the 5th spindle's speed through the special D (in accordance with **M2x774**).	R/W	Decimal	0 ~ 4,294,967,295
5th Spindle Speed Rate	D2x334	Setting the spindle's speed ratio. Ex: When the program speed is S1000 and this special D is 30, then the NC system will execute the spindle speed as 1000 x 30% = 300 RPM.	R/W	Decimal	0 ~ 65,535
5th Spindle Gear Ratio Selection	D2x335	Select in accordance with **N0.1234 ~ N0.1241**. Ex: When this special D is set to 1, the system will set the 5thth spindle to 1st gear, when it is set to 2, it will be in 2nd gear, and so forth.	R/W	Decimal	0 ~ 65,535
6th Spindle Speed	D2x338 D2x339	Write the 6th spindle's speed through the special D (in accordance with **M2x790**).	R/W	Decimal	0 ~ 4,294,967,295
6th Spindle Speed Rate	D2x340	Setting the spindle's speed ratio. Ex: When the program speed is S1000 and this special D is 30, then the NC system will execute the spindle speed as 1000 x 30% = 300 RPM.	R/W	Decimal	0 ~ 65,535
6th Spindle Gear Ratio Selection	D2x341	Select in accordance with **N0.1284 ~ N0.1291**. Ex: When this special D is set to 1, the system will set the 6th spindle to 1st gear, when it is set to 2, it will be in 2nd gear, and so forth.	R/W	Decimal	0 ~ 65,535
7th Spindle Speed	D2x344 D2x345	Write the 7th spindle's speed through the special D (in accordance with **M2x806**).	R/W	Decimal	0 ~ 4,294,967,295
7th Spindle Speed Rate	D2x346	Setting the spindle's speed ratio. Ex: When the program speed is S1000 and this special D is 30, then the NC system will execute the spindle speed as 1000 x 30% = 300 RPM.	R/W	Decimal	0 ~ 65,535
7th Spindle Gear Ratio Selection	D2x347	Select in accordance with **N0.1334 ~ N0.1341**. Ex: When this special D is set to 1, the system will set the 7th spindle to 1st gear, when it is set to 2, it will be in 2nd gear, and so forth.	R/W	Decimal	0 ~ 65,535
8th Spindle Speed	D2x350 D2x351	Write the 8th spindle's speed through the special D (in accordance with **M2x822**).	R/W	Decimal	0 ~ 4,294,967,295
8th Spindle Speed Rate	D2x352	Setting the spindle's speed ratio. Ex: When the program speed is S1000 and this special D is 30, then the NC system will execute the spindle speed as 1000 x 30% = 300 RPM.	R/W	Decimal	0 ~ 65,535

Function Name	Special D	Description	Device	Type	Range
8th Spindle Gear Ratio Selection	D2x353	Select in accordance with **N0.1384 ~ N0.1391**. Ex: When this special D is set to 1, the system will set the 8th spindle to 1st gear, when it is set to 2, it will be in 2nd gear, and so forth.	R/W	Decimal	0 ~ 65,535

B.2.3.2 D49000 to D49899

Function Name	Special D	Description	Device	Type	Range
Complete Process Amount	D49x00 D49x01	Available to set this special D from system interface or MLC.	R/W	Decimal	0 ~ 4,294,967,295
Process Target Amount	D49x02 D49x03	Available to set this special D from system interface or MLC.	R/W	Decimal	0 ~ 4,294,967,295
Total Process Time	D49x04 D49x05	When the system parameter **[N6.032 Process Time Record]** is 1, the system will automatically record the total process time, in units of seconds.	R/W	Decimal	0 ~ 4,294,967,295
Single Process Time	D49x06 D49x07	When the system parameter **[N6.032 Process Time Record]** is 1, the system will automatically record the single process time, in units of seconds.	R/W	Decimal	0 ~ 4,294,967,295

B.2.4 List of special D (System status)

B.2.4.1 D30000 to D38999

B

Function Name	Special D	Description	Device	Type	Range
Pulse Feedback of Spindle 1	D30000	Pulse feedback of spindle 1 connector.	R	Decimal	0 ~ 65,535
Pulse Feedback of Spindle 2	D30001	Pulse feedback of spindle 2 connector.	R	Decimal	0 ~ 65,535
Z Phase Pulse Feedback of Spindle 1	D30002	Z phase pulse feedback of spindle 1 connector.	R	Decimal	0 ~ 65,535
Z Phase Pulse Feedback of Spindle 2	D30003	Z phase pulse feedback of spindle 2 connector.	R	Decimal	0 ~ 65,535
MPG Pulse Feedback	D30004	Pulse feedback of MPG connector.	R	Decimal	0 ~ 65,535
Pulse Output of Spindle 1	D30008	Pulse output of spindle 1 connector.	R	Decimal	0 ~ 65,535
Pulse Output of Spindle 2	D30009	Pulse output of spindle 2 connector.	R	Decimal	0 ~ 65,535
Spindle 1 Pulse Feedback Coordinate	D30174 D30175	Display the pulse feedback coordinate from the Spindle 1 connector. The NC system will calculate the pulse feedback from the Spindle 1 connector as the 1st spindle's feedback coordinate, which based on parameter setting of **[N0.030 ~ N0.032]**. When the [N0.030] set as rotary axis, this special D register will display the position between 0 ~ 359.999.	R	Float	-2,147,483,648 ~ +2,147,483,647
Spindle 2 Pulse Feedback Coordinate	D30176 D30177	Display the pulse feedback coordinate from the Spindle 2 connector. The NC system will calculate the pulse feedback from the Spindle 2 connector as the 2nd spindle's feedback coordinate, which based on parameter setting of **[N0.035 ~ N0.037]**. When the [N0.035] set as rotary axis, this special D register will display the position between 0 ~ 359.999.	R	Float	-2,147,483,648 ~ +2,147,483,647
1st Remote Module Input	D30200 D30201 D30202 D30203	For connected modules that do not include digital input/output (DI/O), the system will set the arrangement order according to the EtherCAT module, and will set the special D module with the special D based on the connection order (such as analog modules). Taking the first module as an example, D30200 corresponds to the first set of values on the module, D30201 corresponds to the second set of values on the module, and so forth.	R	Decimal	0 ~ 65,535
2nd Remote Module Input	D30204 D30205 D30206 D30207		R	Decimal	0 ~ 65,535
3rd Remote Module Input	D30208 D30209 D30210 D30211		R	Decimal	0 ~ 65,535
4th Remote Module Input	D30212 D30213 D30214 D30215		R	Decimal	0 ~ 65,535
5th Remote Module Input	D30216 D30217 D30218 D30219		R	Decimal	0 ~ 65,535

Function Name	Special D	Description	Device	Type	Range
6th Remote Module Input	D30220 D30221 D30222 D30223	For connected modules that do not include digital input/output (DI/O), the system will set the arrangement order according to the EtherCAT module, and will set the special D module with the special D based on the connection order (such as analog modules). Taking the first module as an example, D30200 corresponds to the first set of values on the module, D30201 corresponds to the second set of values on the module, and so forth.	R	Decimal	0 ~ 65,535
7th Remote Module Input	D30224 D30225 D30226 D30227		R	Decimal	0 ~ 65,535
8th Remote Module Input	D30228 D30229 D30230 D30231		R	Decimal	0 ~ 65,535
Torque Feedback of 1st Axis	D30240	Torque feedback for the axis. The axis index is set according to the NC channel axis sequence setting. PDO address 0x6077H Unit: 0.1%	R	Decimal	-32,768 ~ +32,767
Torque Feedback of 2nd Axis	D30241		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 3rd Axis	D30242		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 4th Axis	D30243		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 5th Axis	D30244		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 6th Axis	D30245		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 7th Axis	D30246		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 8th Axis	D30247		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 9th Axis	D30248		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 10th Axis	D30249		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 11th Axis	D30250		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 12th Axis	D30251		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 13th Axis	D30252		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 14th Axis	D30253		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 15th Axis	D30254		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 16th Axis	D30255		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 17th Axis	D30256		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 18th Axis	D30257		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 19th Axis	D30258		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 20th Axis	D30259		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 21th Axis	D30260		R	Decimal	-32,768 ~ +32,767

B

Function Name	Special D	Description	Device	Type	Range
Torque Feedback of 22th Axis	D30261	Torque feedback for the axis. The axis index is set according to the NC channel axis sequence setting. PDO address 0x6077H Unit: 0.1%	R	Decimal	-32,768 ~ +32,767
Torque Feedback of 23th axis	D30262		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 24th Axis	D30263		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 25th Axis	D30264		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 26th Axis	D30265		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 27th Axis	D30266		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 28th Axis	D30267		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 29th Axis	D30268		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 30th Axis	D30269		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 31th Axis	D30270		R	Decimal	-32,768 ~ +32,767
Torque Feedback of 32th Axis	D30271		R	Decimal	-32,768 ~ +32,767
Velocity Feedback of 1st Axis (mm/min)	D30272 D30273	Axis speed feedback for the axis. The axis index is set according to the NC channel axis sequence setting. PDO address 0x606CH Unit: mm/min	R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 2nd Axis (mm/min)	D30274 D30275		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 3rd Axis (mm/min)	D30276 D30277		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 4th Axis (mm/min)	D30278 D30279		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 5th Axis (mm/min)	D30280 D30281		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 6th Axis (mm/min)	D30282 D30283		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 7th Axis (mm/min)	D30284 D30285		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 8th Axis (mm/min)	D30286 D30287		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 9th Axis (mm/min)	D30288 D30289		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 10th Axis (mm/min)	D30290 D30291		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 11th Axis (mm/min)	D30292 D30293		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 12th Axis (mm/min)	D30294 D30295		R	Float	-2,147,483,648 ~ +2,147,483,647

Function Name	Special D	Description	Device	Type	Range
Velocity Feedback of 13th Axis (mm/min)	D30296 D30297	Axis speed feedback for the axis. The axis index is set according to the NC channel axis sequence setting. PDO address 0x606CH Unit: mm/min	R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 14th Axis (mm/min)	D30298 D30299		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 15th Axis (mm/min)	D30300 D30301		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 16th Axis (mm/min)	D30302 D30303		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 17th Axis (mm/min)	D30304 D30305		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 18th Axis (mm/min)	D30306 D30307		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 19th Axis (mm/min)	D30308 D30309		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 20th Axis (mm/min)	D30310 D30311		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 21th Axis (mm/min)	D30312 D30313		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 22th Axis (mm/min)	D30314 D30315		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 23th Axis (mm/min)	D30316 D30317		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 24th Axis (mm/min)	D30318 D30319		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 25th Axis (mm/min)	D30320 D30321		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 26th Axis (mm/min)	D30322 D30323		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 27th Axis (mm/min)	D30324 D30325		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 28th Axis (mm/min)	D30326 D30327		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 29th Axis (mm/min)	D30328 D30329		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 30th Axis (mm/min)	D30330 D30331		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 31th Axis (mm/min)	D30332 D30333		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 32th Axis (mm/min)	D30334 D30335		R	Float	-2,147,483,648 ~ +2,147,483,647

B

Function Name	Special D	Description	Device	Type	Range
Velocity Feedback of 1st Axis (RPM)	D30336 D30337	Axis speed feedback for the axis. The axis index is set according to the NC channel axis sequence setting. PDO address 0x606CH Unit: RPM	R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 2nd Axis (RPM)	D30338 D30339		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 3rd Axis (RPM)	D30340 D30341		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 4th Axis (RPM)	D30342 D30343		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 5th Axis (RPM)	D30344 D30345		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 6th Axis (RPM)	D30346 D30347		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 7th Axis (RPM)	D30348 D30349		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 8th Axis (RPM)	D30350 D30351		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 9th Axis (RPM)	D30352 D30353		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 10th Axis (RPM)	D30354 D30355		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 11th Axis (RPM)	D30356 D30357		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 12th Axis (RPM)	D30358 D30359		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 13th Axis (RPM)	D30360 D30361		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 14th Axis (RPM)	D30362 D30363		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 15th Axis (RPM)	D30364 D30365		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 16th Axis (RPM)	D30366 D30367		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 17th Axis (RPM)	D30368 D30369		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 18th Axis (RPM)	D30360 D30371		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 19th Axis (RPM)	D30372 D30373		R	Float	-2,147,483,648 ~ +2,147,483,647

Function Name	Special D	Description	Device	Type	Range
Velocity Feedback of 20th Axis (RPM)	D30374 D30375	Axis speed feedback for the axis. The axis index is set according to the NC channel axis sequence setting. PDO address 0x606CH Unit: RPM	R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 21th Axis (RPM)	D30376 D30377		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 22th Axis (RPM)	D30378 D30379		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 23th Axis (RPM)	D30380 D30381		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 24th Axis (RPM)	D30382 D30383		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 25th Axis (RPM)	D30384 D30385		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 26th Axis (RPM)	D30386 D30387		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 27th Axis (RPM)	D30388 D30389		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 28th Axis (RPM)	D30390 D30391		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 29th Axis (RPM)	D30392 D30393		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 30th Axis (RPM)	D30394 D30395		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 31th Axis (RPM)	D30396 D30397		R	Float	-2,147,483,648 ~ +2,147,483,647
Velocity Feedback of 32th Axis (RPM)	D30398 D30399		R	Float	-2,147,483,648 ~ +2,147,483,647
Torque Peak of 1st Axis	D30400	The system will monitor and record the maximum axis torque shown in **D30240** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 2nd Axis	D30401	The system will monitor and record the maximum axis torque shown in **D30241** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 3rd Axis	D30402	The system will monitor and record the maximum axis torque shown in **D30242** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 4th Axis	D30403	The system will monitor and record the maximum axis torque shown in **D30243** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295

B

Function Name	Special D	Description	Device	Type	Range
Torque Peak of 5th Axis	D30404	The system will monitor and record the maximum axis torque shown in **D30244** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 6th Axis	D30405	The system will monitor and record the maximum axis torque shown in **D30245** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 7th Axis	D30406	The system will monitor and record the maximum axis torque shown in **D30246** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 8th Axis	D30407	The system will monitor and record the maximum axis torque shown in **D30247** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 9th Axis	D30408	The system will monitor and record the maximum axis torque shown in **D30248** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 10th Axis	D30409	The system will monitor and record the maximum axis torque shown in **D30249** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 11th Axis	D30410	The system will monitor and record the maximum axis torque shown in **D30250** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 12th Axis	D30411	The system will monitor and record the maximum axis torque shown in **D30251** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 13th Axis	D30412	The system will monitor and record the maximum axis torque shown in **D30252** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 14th Axis	D30413	The system will monitor and record the maximum axis torque shown in **D30253** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 15th Axis	D30414	The system will monitor and record the maximum axis torque shown in **D30254** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 16th Axis	D30415	The system will monitor and record the maximum axis torque shown in **D30255** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 17th Axis	D30416	The system will monitor and record the maximum axis torque shown in **D30256** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295

Function Name	Special D	Description	Device	Type	Range
Torque Peak of 18th Axis	D30417	The system will monitor and record the maximum axis torque shown in **D30257** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 19th Axis	D30418	The system will monitor and record the maximum axis torque shown in **D30258** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 20th Axis	D30419	The system will monitor and record the maximum axis torque shown in **D30259** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 21th Axis	D30420	The system will monitor and record the maximum axis torque shown in **D30260** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit:	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 22th Axis	D30421	The system will monitor and record the maximum axis torque shown in **D30261** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 23th Axis	D30422	The system will monitor and record the maximum axis torque shown in **D30262** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 24th Axis	D30423	The system will monitor and record the maximum axis torque shown in **D30263** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 25th Axis	D30424	The system will monitor and record the maximum axis torque shown in **D30264** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 26th Axis	D30425	The system will monitor and record the maximum axis torque shown in **D30265** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 27th Axis	D30426	The system will monitor and record the maximum axis torque shown in **D30266** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 28th Axis	D30427	The system will monitor and record the maximum axis torque shown in **D30267** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 29th Axis	D30428	The system will monitor and record the maximum axis torque shown in **D30268** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 30th Axis	D30429	The system will monitor and record the maximum axis torque shown in **D30269** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295

B

Function Name	Special D	Description	Device	Type	Range
Torque Peak of 31th Axis	D30430	The system will monitor and record the maximum axis torque shown in **D30270** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
Torque Peak of 32th Axis	D30431	The system will monitor and record the maximum axis torque shown in **D30271** during Servo ON. Once the axis enters Servo OFF, these special D will be cleared as well. Unit: 0.1%.	R	Decimal	0 ~ 4,294,967,295
1st Spindle Torque Feedback	D30432	Display the spindle current torque feedback according to the spindle ID setting in the channel. Unit: 0.1%	R	Decimal	-32,768 ~ +32,767
2nd Spindle Torque Feedback	D30433		R	Decimal	-32,768 ~ +32,767
3rd Spindle Torque Feedback	D30434		R	Decimal	-32,768 ~ +32,767
4th Spindle Torque Feedback	D30435		R	Decimal	-32,768 ~ +32,767
5th Spindle Torque Feedback	D30436		R	Decimal	-32,768 ~ +32,767
6th Spindle Torque Feedback	D30437		R	Decimal	-32,768 ~ +32,767
7th Spindle Torque Feedback	D30438		R	Decimal	-32,768 ~ +32,767
8th Spindle Torque Feedback	D30439		R	Decimal	-32,768 ~ +32,767
1st Spindle Torque Feedback Peak	D30440	The NC system will record the maximum torque feedback which showing in the **D30432** when the spindle is enabled. Value cleared when the spindle is disabled. Unit: 0.1%	R	Decimal	-32,768 ~ +32,767
2nd Spindle Torque Feedback Peak	D30441	The NC system will record the maximum torque feedback which showing in the **D30433** when the spindle is enabled. Value cleared when the spindle is disabled. Unit: 0.1%	R	Decimal	-32,768 ~ +32,767
3rd Spindle Torque Feedback Peak	D30442	The NC system will record the maximum torque feedback which showing in the **D30434** when the spindle is enabled. Value cleared when the spindle is disabled. Unit: 0.1%	R	Decimal	-32,768 ~ +32,767
4th Spindle Torque Feedback Peak	D30443	The NC system will record the maximum torque feedback which showing in the **D30435** when the spindle is enabled. Value cleared when the spindle is disabled. Unit: 0.1%	R	Decimal	-32,768 ~ +32,767
5th Spindle Torque Feedback Peak	D30444	The NC system will record the maximum torque feedback which showing in the **D30436** when the spindle is enabled. Value cleared when the spindle is disabled. Unit: 0.1%	R	Decimal	-32,768 ~ +32,767
6th Spindle Torque Feedback Peak	D30445	The NC system will record the maximum torque feedback which showing in the **D30437** when the spindle is enabled. Value cleared when the spindle is disabled. Unit: 0.1%	R	Decimal	-32,768 ~ +32,767

Function Name	Special D	Description	Device	Type	Range
7th Spindle Torque Feedback Peak	D30446	The NC system will record the maximum torque feedback which showing in the **D30438** when the spindle is enabled. Value cleared when the spindle is disabled. Unit: 0.1%	R	Decimal	-32,768 ~ +32,767
8th Spindle Torque Feedback Peak	D30447	The NC system will record the maximum torque feedback which showing in the **D30439** when the spindle is enabled. Value cleared when the spindle is disabled. Unit: 0.1%	R	Decimal	-32,768 ~ +32,767
Current Status of Multi-Z axis	D3x014	This special D will show the Z axes index currently in use when the Multi-Z axis function is enabled. When the value is 12, it means Z1 and Z2 are working. When the value is 123, it means Z1, Z2, and Z3 are working.	R	Decimal	0 ~ 4,294,967,295
Current Coordinate System	D3x016	Shows the coordinate system the system is currently using. This coordinate system can be different according to the channel machine type setting such as standard type and Robot. When the channel set as standard machine type such as milling or lathe, the system will show as below working coordinate system. 1~6: corresponding to G54~G59. 7~262: corresponding to G54 P1~G54 P256. When the channel set as Robot machine type, the system will show as below working coordinate system. 1~6: corresponding to G54~G59.	R	Decimal	Standard 1 ~ 262 Robot 1~6
Current Robot Tool Coordinate System	D3x017	Shows the robot tool coordinate system. 0: Not using tool offset 1 ~ n: corresponds to tool number	R	Decimal	0 ~ 65,535
Speed Command of 1st Spindle	D3x024	When the 1st spindle S code is executed in a program, the command value will be sent in this special D. Unit: RPM	R	Float	-2,147,483,648 ~ +2,147,483,647
1st Spindle Speed Feedback	D3x026	Shows the 1st spindle's speed. The data source is from the spindle's command speed.	R	Float	-2,147,483,648 ~ +2,147,483,647
1st Spindle Actual Degree	D3x028	Shows the 1st spindle's actual degree. The data source is from the spindle's actual degree.	R	Float	-2,147,483,648 ~ +2,147,483,647
Speed Command of 2nd Spindle	D3x030	When the 2nd spindle S code is executed in a program, the command value will be sent in this special D. Unit: RPM	R	Float	-2,147,483,648 ~ +2,147,483,647
2nd Spindle Speed Feedback	D3x032	Shows the 2nd spindle's speed. The data source is from the spindle's command speed.	R	Float	-2,147,483,648 ~ +2,147,483,647
2nd Spindle Actual Degree	D3x034	Shows the 2nd spindle's actual degree. The data source is from the spindle's actual degree.	R	Float	-2,147,483,648 ~ +2,147,483,647
Current Tool Number Tool Magazine 1	D3x036	Current tool number in the tool magazine 1.	R	Decimal	0 ~ 65,535

B

Function Name	Special D	Description	Device	Type	Range
Standby Tool Number Tool Magazine 1	D3x037	Current standby tool number in the tool magazine 1. (The latest T code)	R	Decimal	0 ~ 65,535
Standby Tool Pot Tool Magazine 1	D3x038	Current standby tool pot number in the tool magazine 1.	R	Decimal	0 ~ 65,535
Tool Pot Deviation Tool Magazine 1	D3x039	The deviation between the positions specified for the current tool and command tool in tool magazine 1. Users can determine the rotation direction by whether this value is positive or negative. When the tool magazine rotates forward (**M2x064**) or backward (**M2x065**) during tool exchange, the tool magazine needs to rotate according to the value to compensate for the offset.	R	Decimal	0 ~ 65,535
Current Tool Number Tool Magazine 2	D3x042	Current tool number in the tool magazine 2.	R	Decimal	0 ~ 65,535
Standby Tool Number Tool Magazine 2	D3x043	Current standby tool number in the tool magazine 2. (The latest T code)	R	Decimal	0 ~ 65,535
Standby Tool Pot Tool Magazine 2	D3x044	Current standby tool pot number in the tool magazine 2.	R	Decimal	0 ~ 65,535
Tool Pot Deviation Tool Magazine 2	D3x045	The deviation between the positions specified for the current tool and command tool in tool magazine 2. Users can determine the rotation direction by whether this value is positive or negative. When the tool magazine rotates forward (**M2x072**) or backward (**M2x073**) during tool exchange, the tool magazine needs to rotate according to the value to compensate for the offset.	R	Decimal	0 ~ 65,535
1st M Code Data	D3x048	When the 1st M code is executed in the program (Not including M00, M01, M02, M30, M98, M99), the M code value will be mapped to this register. When the M code is used to call the macro, this special D will keep the previous value.	R	Decimal	0 ~ 4,294,967,295
1st S Code Data	D3x050	When the 1st spindle's 1st S code is executed in the program, the 1st spindle's S code will be mapped to this register. Unit: RPM.	R	Decimal	0 ~ 4,294,967,295
1st T Code Data	D3x052	When the 1st T code is executed in the program, the T code will be mapped to this register. When the 1st T code is used to call the macro, this special D will keep the previous value. This data is related to the tool pot setting of the tool magazine, and the T code will be shown only when the T code value is set within the specified range of tool number for the tool magazine parameter.	R	Decimal	0 ~ 4,294,967,295

Function Name	Special D	Description	Device	Type	Range
Information Monitoring 1 Sort 1	D3x096 D3x097	This special D is for users to monitor the information of system devices according to the variable settings. In order to use this special D, users need to set up [N1.321] for information type and [N1.322] for data sort 1 in advance. *Please check section 6.17 for more details.	R	Decimal	0 ~ 65,535
Information Monitoring 1 Sort 2	D3x098 D3x099	This special D is for users to monitor the information of system devices according to the variable settings. In order to use this special D, users need to set up [N1.321] for information type and [N1.323] for data sort 2 in advance. *Please check section 6.17 for more details.	R	Decimal	0 ~ 65,535
Information Monitoring 1 Sort 3	D3x100 D3x101	This special D is for users to monitor the information of system devices according to the variable settings. In order to use this special D, users need to set up [N1.321] for information type and [N1.324] for data sort 3 in advance. *Please check section 6.17 for more details.	R	Decimal	0 ~ 65,535
Information Monitoring 1 Sort 4	D3x102 D3x103	This special D is for users to monitor the information of system devices according to the variable settings. In order to use this special D, users need to set up [N1.321] for information type and [N1.325] for data sort 4 in advance. *Please check section 6.17 for more details.	R	Decimal	0 ~ 65,535
Information Monitoring 2 Sort 1	D3x104 D3x105	This special D is for users to monitor the information of system devices according to the variable settings. In order to use this special D, users need to set up [N1.326] for information type and [N1.327] for data sort 1 in advance. *Please check section 6.17 for more details.	R	Decimal	0 ~ 65,535
Information Monitoring 2 Sort 2	D3x106 D3x107	This special D is for users to monitor the information of system devices according to the variable settings. In order to use this special D, users need to set up [N1.326] for information type and [N1.328] for data sort 2 in advance. *Please check section 6.17 for more details.	R	Decimal	0 ~ 65,535
Information Monitoring 2 Sort 3	D3x108 D3x109	This special D is for users to monitor the information of system devices according to the variable settings. In order to use this special D, users need to set up [N1.326] for information type and [N1.329] for data sort 3 in advance. *Please check section 6.17 for more details.	R	Decimal	0 ~ 65,535
Information Monitoring 2 Sort 4	D3x110 D3x111	This special D is for users to monitor the information of system devices according to the variable settings. In order to use this special D, users need to set up [N1.326] for information type and [N1.330] for data sort 4 in advance. *Please check section 6.17 for more details.	R	Decimal	0 ~ 65,535

B

Function Name	Special D	Description	Device	Type	Range
NC Variable to MLC 1	D3x128	The system will move data from NC variable #25384 to this special D.	R	Decimal / Float	(Default) -32,768 ~ +32,767 / (N1.010 Bit7=1) -2,147,483,648 ~ +2,147,483,647
NC Variable to MLC 2	D3x129	The system will move data from NC variable #25385 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 3	D3x130	The system will move data from NC variable #25386 to this special D.	R		
NC Variable to MLC 4	D3x131	The system will move data from NC variable #25387 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 5	D3x132	The system will move data from NC variable #25388 to this special D.	R		
NC Variable to MLC 6	D3x133	The system will move data from NC variable #25389 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 7	D3x134	The system will move data from NC variable #25390 to this special D.	R		
NC Variable to MLC 8	D3x135	The system will move data from NC variable #25391 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 9	D3x136	The system will move data from NC variable #25392 to this special D.	R		
NC Variable to MLC 10	D3x137	The system will move data from NC variable #25393 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 11	D3x138	The system will move data from NC variable #25394 to this special D.	R		
NC Variable to MLC 12	D3x139	The system will move data from NC variable #25395 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 13	D3x140	The system will move data from NC variable #25396 to this special D.	R		
NC Variable to MLC 14	D3x141	The system will move data from NC variable #25397 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		

Function Name	Special D	Description	Device	Type	Range
NC Variable to MLC 15	D3x142	The system will move data from NC variable #25398 to this special D.	R	Decimal / Float	(Default) -32,768 ~ +32,767 / (N1.010 Bit7=1) -2,147,483,648 ~ +2,147,483,647
NC Variable to MLC 16	D3x143	The system will move data from NC variable #25399 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 17	D3x144	The system will move data from NC variable #25400 to this special D.	R		
NC Variable to MLC 18	D3x145	The system will move data from NC variable #25401 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 19	D3x146	The system will move data from NC variable #25402 to this special D.	R		
NC Variable to MLC 20	D3x147	The system will move data from NC variable #25403 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 21	D3x148	The system will move data from NC variable #25404 to this special D.	R		
NC Variable to MLC 22	D3x149	The system will move data from NC variable #25405 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 23	D3x150	The system will move data from NC variable #25406 to this special D.	R		
NC Variable to MLC 24	D3x151	The system will move data from NC variable #25407 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 25	D3x152	The system will move data from NC variable #25408 to this special D.	R		
NC Variable to MLC 26	D3x153	The system will move data from NC variable #25409 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 27	D3x154	The system will move data from NC variable #25410 to this special D.	R		
NC Variable to MLC 28	D3x155	The system will move data from NC variable #25411 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		

B

Function Name	Special D	Description	Device	Type	Range
NC Variable to MLC 29	D3x156	The system will move data from NC variable #25412 to this special D.	R	Decimal / Float	(Default) -32,768 ~ +32,767 / (N1.010 Bit7=1) -2,147,483,648 ~ +2,147,483,647
NC Variable to MLC 30	D3x157	The system will move data from NC variable #25413 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 31	D3x158	The system will move data from NC variable #25414 to this special D.	R		
NC Variable to MLC 32	D3x159	The system will move data from NC variable #25415 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 33	D3x160	The system will move data from NC variable #25416 to this special D.	R		
NC Variable to MLC 34	D3x161	The system will move data from NC variable #25417 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 35	D3x162	The system will move data from NC variable #25418 to this special D.	R		
NC Variable to MLC 36	D3x163	The system will move data from NC variable #25419 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 37	D3x164	The system will move data from NC variable #25420 to this special D.	R		
NC Variable to MLC 38	D3x165	The system will move data from NC variable #25421 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 39	D3x166	The system will move data from NC variable #25422 to this special D.	R		
NC Variable to MLC 40	D3x167	The system will move data from NC variable #25423 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 41	D3x168	The system will move data from NC variable #25424 to this special D.	R		
NC Variable to MLC 42	D3x169	The system will move data from NC variable #25425 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		

Function Name	Special D	Description	Device	Type	Range
NC Variable to MLC 43	D3x170	The system will move data from NC variable #25426 to this special D.	R	Decimal / Float	(Default) -32,768 ~ +32,767 / (N1.010 Bit7=1) -2,147,483,648 ~ +2,147,483,647
NC Variable to MLC 44	D3x171	The system will move data from NC variable #25427 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 45	D3x172	The system will move data from NC variable #25428 to this special D.	R		
NC Variable to MLC 46	D3x173	The system will move data from NC variable #25429 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 47	D3x174	The system will move data from NC variable #25430 to this special D.	R		
NC Variable to MLC 48	D3x175	The system will move data from NC variable #25431 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 49	D3x176	The system will move data from NC variable #25432 to this special D.	R		
NC Variable to MLC 50	D3x177	The system will move data from NC variable #25433 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 51	D3x178	The system will move data from NC variable #25434 to this special D.	R		
NC Variable to MLC 52	D3x179	The system will move data from NC variable #25435 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 53	D3x180	The system will move data from NC variable #25436 to this special D.	R		
NC Variable to MLC 54	D3x181	The system will move data from NC variable #25437 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 55	D3x182	The system will move data from NC variable #25438 to this special D.	R		
NC Variable to MLC 56	D3x183	The system will move data from NC variable #25439 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		

B

Function Name	Special D	Description	Device	Type	Range
NC Variable to MLC 57	D3x184	The system will move data from NC variable #25440 to this special D.	R	Decimal / Float	(Default) -32,768 ~ +32,767 / (N1.010 Bit7=1) -2,147,483,648 ~ +2,147,483,647
NC Variable to MLC 58	D3x185	The system will move data from NC variable #25441 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 59	D3x186	The system will move data from NC variable #25442 to this special D.	R		
NC Variable to MLC 60	D3x187	The system will move data from NC variable #25443 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 61	D3x188	The system will move data from NC variable #25444 to this special D.	R		
NC Variable to MLC 62	D3x189	The system will move data from NC variable #25445 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 63	D3x190	The system will move data from NC variable #25446 to this special D.	R		
NC Variable to MLC 64	D3x191	The system will move data from NC variable #25447 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 65	D3x192	The system will move data from NC variable #25448 to this special D.	R		
NC Variable to MLC 66	D3x193	The system will move data from NC variable #25449 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 67	D3x194	The system will move data from NC variable #25450 to this special D.	R		
NC Variable to MLC 68	D3x195	The system will move data from NC variable #25451 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 69	D3x196	The system will move data from NC variable #25452 to this special D.	R		
NC Variable to MLC 70	D3x197	The system will move data from NC variable #25453 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		

Function Name	Special D	Description	Device	Type	Range
NC Variable to MLC 71	D3x198	The system will move data from NC variable #25454 to this special D.	R	Decimal / Float	(Default) -32,768 ~ +32,767 / (N1.010 Bit7=1) -2,147,483,648 ~ +2,147,483,647
NC Variable to MLC 72	D3x199	The system will move data from NC variable #25455 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 73	D3x200	The system will move data from NC variable #25456 to this special D.	R		
NC Variable to MLC 74	D3x201	The system will move data from NC variable #25457 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 75	D3x202	The system will move data from NC variable #25458 to this special D.	R		
NC Variable to MLC 76	D3x203	The system will move data from NC variable #25459 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 77	D3x204	The system will move data from NC variable #25460 to this special D.	R		
NC Variable to MLC 78	D3x205	The system will move data from NC variable #25461 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 79	D3x206	The system will move data from NC variable #25462 to this special D.	R		
NC Variable to MLC 80	D3x207	The system will move data from NC variable #25463 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 81	D3x208	The system will move data from NC variable #25464 to this special D.	R		
NC Variable to MLC 82	D3x209	The system will move data from NC variable #25465 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 83	D3x210	The system will move data from NC variable #25466 to this special D.	R		
NC Variable to MLC 84	D3x211	The system will move data from NC variable #25467 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		

B

Function Name	Special D	Description	Device	Type	Range
NC Variable to MLC 85	D3x212	The system will move data from NC variable #25468 to this special D.	R	Decimal / Float	(Default) -32,768 ~ +32,767 / (N1.010 Bit7=1) -2,147,483,648 ~ +2,147,483,647
NC Variable to MLC 86	D3x213	The system will move data from NC variable #25469 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 87	D3x214	The system will move data from NC variable #25470 to this special D.	R		
NC Variable to MLC 88	D3x215	The system will move data from NC variable #25471 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 89	D3x216	The system will move data from NC variable #25472 to this special D.	R		
NC Variable to MLC 90	D3x217	The system will move data from NC variable #25473 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 91	D3x218	The system will move data from NC variable #25474 to this special D.	R		
NC Variable to MLC 92	D3x219	The system will move data from NC variable #25475 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 93	D3x220	The system will move data from NC variable #25476 to this special D.	R		
NC Variable to MLC 94	D3x221	The system will move data from NC variable #25477 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 95	D3x222	The system will move data from NC variable #25478 to this special D.	R		
NC Variable to MLC 96	D3x223	The system will move data from NC variable #25479 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 97	D3x224	The system will move data from NC variable #25480 to this special D.	R		
NC Variable to MLC 98	D3x225	The system will move data from NC variable #25481 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		

Function Name	Special D	Description	Device	Type	Range
NC Variable to MLC 99	D3x226	The system will move data from NC variable #25482 to this special D.	R	Decimal / Float	(Default) -32,768 ~ +32,767 / (N1.010 Bit7=1) -2,147,483,648 ~ +2,147,483,647
NC Variable to MLC 100	D3x227	The system will move data from NC variable #25483 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 101	D3x228	The system will move data from NC variable #25484 to this special D.	R		
NC Variable to MLC 102	D3x229	The system will move data from NC variable #25485 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 103	D3x230	The system will move data from NC variable #25486 to this special D.	R		
NC Variable to MLC 104	D3x231	The system will move data from NC variable #25487 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 105	D3x232	The system will move data from NC variable #25488 to this special D.	R		
NC Variable to MLC 106	D3x233	The system will move data from NC variable #25489 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 107	D3x234	The system will move data from NC variable #25490 to this special D.	R		
NC Variable to MLC 108	D3x235	The system will move data from NC variable #25491 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 109	D3x236	The system will move data from NC variable #25492 to this special D.	R		
NC Variable to MLC 110	D3x237	The system will move data from NC variable #25493 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 111	D3x238	The system will move data from NC variable #25494 to this special D.	R		
NC Variable to MLC 112	D3x239	The system will move data from NC variable #25495 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		

B

Function Name	Special D	Description	Device	Type	Range
NC Variable to MLC 113	D3x240	The system will move data from NC variable #25496 to this special D.	R	Decimal / Float	(Default) -32,768 ~ +32,767 / (N1.010 Bit7=1) -2,147,483,648 ~ +2,147,483,647
NC Variable to MLC 114	D3x241	The system will move data from NC variable #25497 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 115	D3x242	The system will move data from NC variable #25498 to this special D.	R		
NC Variable to MLC 116	D3x243	The system will move data from NC variable #25499 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 117	D3x244	The system will move data from NC variable #25500 to this special D.	R		
NC Variable to MLC 118	D3x245	The system will move data from NC variable #25501 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 119	D3x246	The system will move data from NC variable #25502 to this special D.	R		
NC Variable to MLC 120	D3x247	The system will move data from NC variable #25503 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 121	D3x248	The system will move data from NC variable #25504 to this special D.	R		
NC Variable to MLC 122	D3x249	The system will move data from NC variable #25505 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 123	D3x250	The system will move data from NC variable #25506 to this special D.	R		
NC Variable to MLC 124	D3x251	The system will move data from NC variable #25507 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
NC Variable to MLC 125	D3x252	The system will move data from NC variable #25508 to this special D.	R		
NC Variable to MLC 126	D3x253	The system will move data from NC variable #25509 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		

Function Name	Special D	Description	Device	Type	Range
NC Variable to MLC 127	D3x254	The system will move data from NC variable #25510 to this special D.	R	Decimal / Float	(Default) -32,768 ~ +32,767 / (N1.010 Bit7=1) -2,147,483,648 ~ +2,147,483,647
NC Variable to MLC 128	D3x255	The system will move data from NC variable #25511 to this special D. *Both this special D and the variable # are NOT available to use when using MLC floating mode **[N1.010 Bit7=1]**. Otherwise, the system will return error 0x235.	R		
Look Ahead Remaining Command	D3x320 D3x321	When the system is executing an NC program, it will preview the current program and planning path, and the remaining number of previewed and executable blocks will be written to this special D.	R	Decimal	0 ~ 4,294,967,295
Rigid Tapping Max Error of 1st Spindle	D3x350 D3x351	The system will record the maximum absolute difference distance between the feed axis feedback and spindle feedback when the 1st spindle executes the tapping function. This special D will be reset prior to every time the tapping function is triggered. Unit: mm	R	Float	-2,147,483,648 ~ +2,147,483,647
Rigid Tapping Max Error of 2nd Spindle	D3x352 D3x353	The system will record the maximum absolute difference distance between the feed axis feedback and spindle feedback when the 2nd spindle executes the tapping function. This special D will be reset prior to every time the tapping function is triggered. Unit: mm	R	Float	-2,147,483,648 ~ +2,147,483,647
Target Feed of 1st Spindle	D3x354 D3x355	Target feed movement of the 1st spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Target Feed of 2nd Spindle	D3x356 D3x357	Target feed movement of the 2nd spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Actual Feed of 1st Spindle	D3x358 D3x359	Actual feed movement of the 1st spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Actual Feed of 2nd Spindle	D3x360 D3x361	Actual feed movement of the 2nd spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Speed Command of 3rd Spindle	D3x362 D3x363	When the 3rd spindle S code is executed in a program, the command value will be sent in this special D. Unit: RPM	R	Float	-2,147,483,648 ~ +2,147,483,647
3rd Spindle Speed Feedback	D3x364 D3x365	Shows the 3rd spindle's speed. The data source is from the spindle's command speed.	R	Float	-2,147,483,648 ~ +2,147,483,647
3rd Spindle Actual Degree	D3x366 D3x367	Shows the 3rd spindle's actual degree. The data source is from the spindle's actual degree.	R	Float	-2,147,483,648 ~ +2,147,483,647

B

Function Name	Special D	Description	Device	Type	Range
Rigid Tapping Max Error of 3rd Spindle	D3x368 D3x369	The system will record the maximum absolute difference distance between the feed axis feedback and spindle feedback when the 3rd spindle executes the tapping function. This special D will be reset prior to every time the tapping function is triggered. Unit: mm	R	Float	-2,147,483,648 ~ +2,147,483,647
Target Feed of 3rd Spindle	D3x370 D3x371	Target feed movement of the 3rd spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Actual Feed of 3rd Spindle	D3x372 D3x373	Actual feed movement of the 3rd spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Speed Command of 4th Spindle	D3x374 D3x375	When the 4th spindle S code is executed in a program, the command value will be sent in this special D. Unit: RPM	R	Float	-2,147,483,648 ~ +2,147,483,647
4th Spindle Speed Feedback	D3x376 D3x377	Shows the 4th spindle's speed. The data source is from the spindle's command speed.	R	Float	-2,147,483,648 ~ +2,147,483,647
4th Spindle Actual Degree	D3x378 D3x379	Shows the 4th spindle's actual degree. The data source is from the spindle's actual degree.	R	Float	-2,147,483,648 ~ +2,147,483,647
Rigid Tapping Max Error of 4th Spindle	D3x380 D3x381	The system will record the maximum absolute difference distance between the feed axis feedback and spindle feedback when the 4th spindle executes the tapping function. This special D will be reset prior to every time the tapping function is triggered. Unit: mm	R	Float	-2,147,483,648 ~ +2,147,483,647
Target Feed of 4th Spindle	D3x382 D3x383	Target feed movement of the 4th spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Actual Feed of 4th Spindle	D3x384 D3x385	Actual feed movement of the 4th spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Speed Command of 5th Spindle	D3x386 D3x387	When the 5th spindle S code is executed in a program, the command value will be sent in this special D. Unit: RPM	R	Float	-2,147,483,648 ~ +2,147,483,647
5th Spindle Speed Feedback	D3x388 D3x389	Shows the 5th spindle's speed. The data source is from the spindle's command speed.	R	Float	-2,147,483,648 ~ +2,147,483,647
5th Spindle Actual Degree	D3x390 D3x391	Shows the 5th spindle's actual degree. The data source is from the spindle's actual degree.	R	Float	-2,147,483,648 ~ +2,147,483,647
Rigid Tapping Max Error of 5th Spindle	D3x392 D3x393	The system will record the maximum absolute difference distance between the feed axis feedback and spindle feedback when the 5th spindle executes the tapping function. This special D will be reset prior to every time the tapping function is triggered. Unit: mm	R	Float	-2,147,483,648 ~ +2,147,483,647
Target Feed of 5th Spindle	D3x394 D3x395	Target feed movement of the 5th spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647

Function Name	Special D	Description	Device	Type	Range
Actual Feed of 5th Spindle	D3x396 D3x397	Actual feed movement of the 5th spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Speed Command of 6th Spindle	D3x398 D3x399	When the 6th spindle S code is executed in a program, the command value will be sent in this special D. Unit: RPM	R	Float	-2,147,483,648 ~ +2,147,483,647
6th Spindle Speed Feedback	D3x400 D3x401	Shows the 6th spindle's speed. The data source is from the spindle's command speed.	R	Float	-2,147,483,648 ~ +2,147,483,647
6th Spindle Actual Degree	D3x402 D3x403	Shows the 6th spindle's actual degree. The data source is from the spindle's actual degree.	R	Float	-2,147,483,648 ~ +2,147,483,647
Rigid Tapping Max Error of 6th Spindle	D3x404 D3x405	The system will record the maximum absolute difference distance between the feed axis feedback and spindle feedback when the 6th spindle executes the tapping function. This special D will be reset prior to every time the tapping function is triggered. Unit: mm	R	Float	-2,147,483,648 ~ +2,147,483,647
Target Feed of 6th Spindle	D3x406 D3x407	Target feed movement of the 6th spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Actual Feed of 6th Spindle	D3x408 D3x409	Actual feed movement of the 6th spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Speed Command of 7th Spindle	D3x410 D3x411	When the 7th spindle S code is executed in a program, the command value will be sent in this special D. Unit: RPM	R	Float	-2,147,483,648 ~ +2,147,483,647
7th Spindle Speed Feedback	D3x412 D3x413	Shows the 7th spindle's speed. The data source is from the spindle's command speed.	R	Float	-2,147,483,648 ~ +2,147,483,647
7th Spindle Actual Degree	D3x414 D3x415	Shows the 7th spindle's actual degree. The data source is from the spindle's actual degree.	R	Float	-2,147,483,648 ~ +2,147,483,647
Rigid Tapping Max Error of 7th Spindle	D3x416 D3x417	The system will record the maximum absolute difference distance between the feed axis feedback and spindle feedback when the 7th spindle executes the tapping function. This special D will be reset prior to every time the tapping function is triggered. Unit: mm	R	Float	-2,147,483,648 ~ +2,147,483,647
Target Feed of 7th Spindle	D3x418 D3x419	Target feed movement of the 7th spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Actual Feed of 7th Spindle	D3x420 D3x421	Actual feed movement of the 7th spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Speed Command of 8th Spindle	D3x422 D3x423	When the 8th spindle S code is executed in a program, the command value will be sent in this special D. Unit: RPM	R	Float	-2,147,483,648 ~ +2,147,483,647
8th Spindle Speed Feedback	D3x424 D3x425	Shows the 8th spindle's speed. The data source is from the spindle's command speed.	R	Float	-2,147,483,648 ~ +2,147,483,647

B

Function Name	Special D	Description	Device	Type	Range
8th Spindle Actual Degree	D3x426 D3x427	Shows the 8th spindle's actual degree. The data source is from the spindle's actual degree.	R	Float	-2,147,483,648 ~ +2,147,483,647
Rigid Tapping Max Error of 8th Spindle	D3x428 D3x429	The system will record the maximum absolute difference distance between the feed axis feedback and spindle feedback when the 8th spindle executes the tapping function. This special D will be reset prior to every time the tapping function is triggered. Unit: mm	R	Float	-2,147,483,648 ~ +2,147,483,647
Target Feed of 8th Spindle	D3x430 D3x431	Target feed movement of the 8th spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647
Actual Feed of 8th Spindle	D3x432 D3x433	Actual feed movement of the 8th spindle in each revolution. Unit: mm/rev.	R	Float	-2,147,483,648 ~ +2,147,483,647

B.3 CNC Special # Variable

This appendix provides functional mapping tables of # variables in the NC 5 controller system. Variables are independently working in each channel of NC, mainly used in operation, read and system setting of NC program.

System # variable corresponding table define as following:

Variable	2	Description	Type
#0	1	system variable of value is null (NULL)	R/-/- 3

1. # variable name, beginning as # and number from 0 to 223999. Number after 20000 are system variables or reserved variables.
2. # variable function description.
3. # variable operation attribute and its attribute description:

   R: # variable read only.

   W: # variable writable.

   ●: When NC system execute this # variable, it will stop look-ahead and block preview which ensure the system can obtain the current state of the NC system.

B.3.1 Constant Variable

B.3.1.1 Null Variable (#0)

<table>
<tr><th>Variable</th><th>Description</th><th>Type</th></tr>
<tr><td>#0</td><td>A null and read only variable (Null).
Users can move #0 value into specified variable, and then set this specified variable as null. (The value will display as NULL).
Null (NULL) and 0 is different in usage, the following table describes the differences when the #100 is given in NULL or 0:
<table>
<tr><th rowspan="2">Judgement</th><th colspan="2">Value of #100</th></tr>
<tr><th>NULL</th><th>0</th></tr>
<tr><td>#100 == #0</td><td>True</td><td>False</td></tr>
<tr><td>#100 != #0</td><td>False</td><td>True</td></tr>
<tr><td>#100 >= #0</td><td>True</td><td>True</td></tr>
<tr><td>#100 <= #0</td><td>True</td><td>True</td></tr>
<tr><td>#100 > 0</td><td>False</td><td>False</td></tr>
<tr><td>#100 < 0</td><td>False</td><td>False</td></tr>
<tr><td>#100 != 0</td><td>True</td><td>False</td></tr>
<tr><td>G90X99Y#100</td><td>G90 X99</td><td>G90 X99 Y0</td></tr>
</table>
</td><td>R/-/-</td></tr>
</table>

B.3.1.2 Constant Variable (#20000~#20012)

B

Variable	Description	Type
#20000	$e \cong 2.718281828$	R/-/-
#20001	$\log_2 e \cong 1.442695041$	R/-/-
#20002	$\log_{10} e \cong 0.434294482$	R/-/-
#20003	$\ln 2 \cong 0.693147181$	R/-/-
#20004	$\ln 10 \cong 2.302585093$	R/-/-
#20005	$\pi \cong 3.141592654$	R/-/-
#20006	$\frac{\pi}{2} \cong 1.570796327$	R/-/-
#20007	$\frac{\pi}{4} \cong 0.785398163$	R/-/-
#20008	$\frac{1}{\pi} \cong 0.318309886$	R/-/-
#20009	$\frac{2}{\pi} \cong 0.636619772$	R/-/-
#20010	$\frac{2}{\sqrt{\pi}} \cong 1.128379167$	R/-/-
#20011	$\sqrt{2} \cong 1.414213562$	R/-/-
#20012	$\frac{1}{\sqrt{\pi}} \cong 0.707106781$	R/-/-

B.3.2 Alarm Variable

B.3.2.1 Macro Defined Alarm (#20020)

Variable	Description	Type
#20020	Macro defined alarm (MR) When non-zero value is written into this variable, the system will display Macro defined alarm on the controller screen. Value range: from 0 to 32767. If the value is not in the range, the system will return alarm 0x0244. The error message from MR1 to MR1000 are configured in the **[CNCSoft]** – **[DOPSoft]**. For example: #20020 = 100; the system displays MR100.	R/W/●

B.3.3 Field Special Variables

B.3.3.1 User-Defined Coordinate (#20021)

Variable	Description	Type
#20021	Special for woodworking machine, used for displaying the working coordinate of the multi-channel. *Need to be used with file sequence function	R/-/●

B.3.3.2 Multi-Z Axis Synchronize (#20022)

Variable	Description	Type
#20022	Slave axes number shielding of multi-Z axis. This value is associated with D3X014.	R/-/●

B

B.3.4 NC System Status

B.3.4.1 Servo Parameter Read and Write (#20023, #20024)

Variable	Description	Type
#20023	This variable store the servo parameter which return from the slave parameters reading instruction. Servo parameters reading instruction includes G10 L40, L41, L45 and L46. For example: After executing G10 L40 I3 P4 D5, this variable will store the value of P4-5 from the servo drive of address number 3. After executing G10 L45 I3 P4 D5, this variable will store the value of [OD-code 4] [Sub OD code 5] from the slave device of address number 3.	R/-/●
#20024	This variable store the execution result (error code) after executing the slave parameters reading instruction. Servo parameters reading instruction includes G10 L40, L41, L45 and L46. For example: After executing G10 L40 I3 P4 D5, this variable will store the execution result (error code). When executing G10 L45 I3 P4 D5, this variable will store the execution result (error code).	R/-/●

B.3.4.2 System Time (#20025, #20026)

Variable	Description	Type
#20025	System time: YYMMDD For example: controller system time is 2023/04/17, the value of #20025 is 230417	R/-/●
#20026	System time: hhmmss For example: controller system time is 17:30:01, the value of #20026 is 173001	R/-/●

B.3.4.3 NC System Mode (#20300~#20330)

Variable	Function description (Milling machine type G code)	Type
#20300	G code group [0] mode: Temporary G command information	R/-/-
#20301	G code group [1] mode: Interpolation mode (G00~G03)	R/-/-
#20302	G code group [2] mode: Plane selection (G17~G19)	R/-/-
#20303	G code group [3] mode: Absolute/ incremental instruction (G90, G91).	R/-/-
#20304	G code group [4] mode: Procedure check (G22, G23).	R/-/-
#20305	G code group [5] mode: Feeding mode G94, G95.	R/-/-
#20306	G code group [6] mode: Metric/ inch mode (G20, G21).	R/-/-
#20307	G code group [7] mode: Tool radius compensation (G40, G41, G42).	R/-/-
#20308	G code group [8] mode: Tool length compensation (G43, G44, G49).	R/-/-
#20309	G code group [9] mode: Circulation instruction (G80).	R/-/-
#20310	G code group [10] mode: Drilling return mode (G98, G99).	R/-/-
#20311	G code group [11] mode: Ratio mode (G50, G51).	R/-/-
#20312	G code group [12] mode: Workpiece coordinate (G54~G59).	R/-/-
#20313	G code group [13] mode: Cutting mode (G61, G64).	R/-/-
#20314	G code group [14] mode: Macro call (G66, G67).	R/-/-
#20315	G code group [15] mode: Coordinate rotation (G68, G69).	R/-/-
#20316	G code group [16] mode: Polar coordinate instruction (G15, G16).	R/-/-
#20317	G code group [17] mode: Cut speed (G96, G97).	R/-/-
#20318	G code group [18] mode: Mirror function (G24, G25).	R/-/-
#20319 …#20330	System reserved. Do not use this sector variable.	-/-/-
#20331	G code group [31] mode: Extend work coordinate (G54 P1-P64).	R/-/-

B.3.4.4 NC Command Status (#20400~#20411)

Variable NO.	Function description	Type
#20400	Current execution feed rate F. This variable records the last F command before the specified search break block when executing breakpoint search function.	R/-/-
#20401	Current tool compensation H value This variable records the last H command before the specified search break block when executing breakpoint search function.	R/-/-
#20402	Current tool compensation D value This variable records the last D command before the specified search break block when executing breakpoint search function.	R/-/-
#20403	Current tool number T value This variable records the last T command before the specified search break block when executing breakpoint search function.	R/-/-
#20404	1st spindle current program speed S.	R/-/●
#20405	2nd spindle current program speed S.	R/-/●
#20406	3rd spindle current program speed S.	R/-/●
#20407	4th spindle current program speed S.	R/-/●
#20408 … #20411	System reserved. Do not use variables in this range.	R/-/●

B.3.4.5 Coordinate Information (#21000~#21079)

	Machine Coordinate	Absolute Coordinate	Target Position	G31 Machine	G31 Absolute	Axis Compensate Feedback	Type
1st axis (X)	#21000	#21016	#21032	#21048	#21064	#21080	R/-/●
2nd axis (Y)	#21001	#21017	#21033	#21049	#21065	#21081	R/-/●
3rd axis (Z)	#21002	#21018	#21034	#21050	#21066	#21082	R/-/●
4th axis (A)	#21003	#21019	#21035	#21051	#21067	#21083	R/-/●
5th axis (B)	#21004	#21020	#21036	#21052	#21068	#21084	R/-/●
6th axis (C)	#21005	#21021	#21037	#21053	#21069	#21085	R/-/●
7th axis (U)	#21006	#21022	#21038	#21054	#21070	#21086	R/-/●
8th axis (V)	#21007	#21023	#21039	#21055	#21071	#21087	R/-/●
9th axis (W)	#21008	#21024	#21040	#21056	#21072	#21088	R/-/●
10th axis	#21009	#21025	#21041	#21057	#21073	#21089	R/-/●
11th axis	#21010	#21026	#21042	#21058	#21074	#21090	R/-/●
12th axis	#21011	#21027	#21043	#21059	#21075	#21091	R/-/●
13th axis	#21012	#21028	#21044	#21060	#21076	#21092	R/-/●
14th axis	#21013	#21029	#21045	#21061	#21077	#21093	R/-/●
15th axis	#21014	#21030	#21046	#21062	#21078	#21094	R/-/●
16th axis	#21015	#21031	#21047	#21063	#21079	#21095	R/-/●

B

B.3.4.6 G54-G59 Workpiece Coordinate (#22000~#23183)

	Offset Position	G54	G55	G56	G57	G58	G59	Type
1st axis (X)	#22000	#22002	#22003	#22004	#22005	#22006	#22007	R/W/●
2nd axis (Y)	#22078	#22080	#22081	#22082	#22083	#22084	#22085	R/W/●
3rd axis (Z)	#22156	#22158	#22159	#22160	#22161	#22162	#22163	R/W/●
4th axis (A)	#22234	#22236	#22237	#22238	#22239	#22240	#22241	R/W/●
5th axis (B)	#22312	#22314	#22315	#22316	#22317	#22318	#22319	R/W/●
6th axis (C)	#22390	#22392	#22393	#22394	#22395	#22396	#22397	R/W/●
7th axis (U)	#22468	#22470	#22471	#22472	#22473	#22474	#22475	R/W/●
8th axis (V)	#22546	#22548	#22549	#22550	#22551	#22552	#22553	R/W/●
9th axis(W)	#22624	#22626	#22627	#22628	#22629	#22630	#22631	R/W/●
10th axis	#22702	#22704	#22705	#22706	#22707	#22708	#22709	R/W/●
11th axis	#22780	#22782	#22783	#22784	#22785	#22786	#22787	R/W/●
12th axis	#22858	#22860	#22861	#22862	#22863	#22864	#22865	R/W/●
13th axis	#22936	#22938	#22939	#22940	#22941	#22942	#22943	R/W/●
14th axis	#23014	#23016	#23017	#23018	#23019	#23020	#23021	R/W/●
15th axis	#23092	#23094	#23095	#23096	#23097	#23098	#23099	R/W/●
16th axis	#23170	#23172	#23173	#23174	#23175	#23176	#23177	R/W/●

	G54 Offset Position	G55 Offset Position	G56 Offset Position	G57 Offset Position	G58 Offset Position	G59 Offset Position	Type
1st axis (X)	#22008	#22009	#22010	#22011	#22012	#22013	R/W/●
2nd axis (Y)	#22086	#22087	#22088	#22089	#22090	#22091	R/W/●
3rd axis (Z)	#22164	#22165	#22166	#22167	#22168	#22169	R/W/●
4th axis (A)	#22242	#22243	#22244	#22245	#22246	#22247	R/W/●
5th axis (B)	#22320	#22321	#22322	#22323	#22324	#22325	R/W/●
6th axis (C)	#22398	#22399	#22400	#22401	#22402	#22403	R/W/●
7th axis (U)	#22476	#22477	#22478	#22479	#22480	#22481	R/W/●
8th axis (V)	#22554	#22555	#22556	#22557	#22558	#22559	R/W/●
9th axis(W)	#22632	#22633	#22634	#22635	#22636	#22637	R/W/●
10th axis	#22710	#22711	#22712	#22713	#22714	#22715	R/W/●
11th axis	#22788	#22789	#22790	#22791	#22792	#22793	R/W/●
12th axis	#22866	#22867	#22868	#22869	#22870	#22871	R/W/●
13th axis	#22944	#22945	#22946	#22947	#22948	#22949	R/W/●
14th axis	#23022	#23023	#23024	#23025	#23026	#23027	R/W/●
15th axis	#23100	#23101	#23102	#23103	#23104	#23105	R/W/●
16th axis	#23178	#23179	#23180	#23181	#23182	#23183	R/W/●

B.3.5 Tool Management and Breakpoints

B.3.5.1 Tool Magazine (#24001~#24004)

B

Variable	Function description	Type
#24001	Activated tool number from the 1st tool magazine.	R/-/●
#24002	Activated tool number from the 2nd tool magazine.	R/-/●
#24003	Tool exchange number of the 1st tool magazine. In the 1st tool magazine, after search the [#24003 specified number] and get the current tool slot, user can exchange the [#24001 activated tool number] and the [#24003 Tool exchange number] and then the tool number in the tool slot will be changed automatically. if #24003 specified tool number is not in the tool magazine, the system will not exchange the tool number.	R/W/●
#24004	Tool exchange number of the 2nd tool magazine. In the 2nd tool magazine, after search the [#24004 specified number] and get the current tool slot, user can exchange the [#24002 activated tool number] and the [#24004 Tool exchange number] and then the tool number in the tool slot will be changed automatically. if #240044 specified tool number is not in the tool magazine, the system will not exchange the tool number.	R/W/●

B.3.5.2 Breakpoint Search (#24039~#24095)

After executing the breakpoint row-searching or label execution, the NC system will directed current line number to the specified breakpoint, and the status information of M, S, T, F and axis coordinates are saved as following table:

Variable	Function Description	Type
#24042 … #24076	Last M code record before breakpoint. When using breakpoint search function, those used M code before breakpoint line number or breakpoint label will record in #24042 to #24076 variables. * If there are more than 35 M codes are used before the breakpoint line number, the system retains only the last 35 sets of M codes based on the first-in-first-out principle. * If less than 35 M codes are used before the breakpoint line number, the corresponding variable will be NULL.	R/-/-
#24077	Last S code record before breakpoint. When using breakpoint search function, the last S code before breakpoint line number or breakpoint label will record in #24077 variable. *If there are no S code has been used before the breakpoint line number, this variable will be null (NULL).	R/-/-
#24078	Last T code record before breakpoint. When using breakpoint search function, those used T code before breakpoint line number or breakpoint label will record in #24078 to #24079 variables. * If there are more than 2 T codes are used before the breakpoint line number, the system retains only the last 2 sets of T codes based on the first-in-first-out principle. * If less than 2 T codes are used before the breakpoint line number, the corresponding variable will be NULL.	R/-/-
#24079		R/-/-

Variable	Function description		Type
#24080	1st axis (X)	Target position of each axis record before breakpoint line. When using breakpoint searching function, the target position before breakpoint line or breakpoint label of each axis will be store in #24080 to #24095 variables. *If the axis is not used before the breakpoint line number, the corresponding variable will be null (NULL).	R/-/-
#24081	2nd axis (Y)		R/-/-
#24082	3rd axis (Z)		R/-/-
#24083	4th axis (A)		R/-/-
#24084	5th axis (B)		R/-/-
#24085	6th axis (C)		R/-/-
#24086	7th axis (U)		R/-/-
#24087	8th axis (V)		R/-/-
#24088	9th axis (W)		R/-/-
#24089	10th axis		R/-/-
#24090	11th axis		R/-/-
#24091	12th axis		R/-/-
#24092	13th axis		R/-/-
#24093	14th axis		R/-/-
#24094	15th axis		R/-/-
#24095	16th axis		R/-/-
#24100	Multiple T code record before breakpoint line. When using breakpoint searching function, the T code command before breakpoint line or breakpoint label will be store in #24100 to #24103 variables. Maximum 4 sets of T code will be recorded. *If the multiple T code is not used before the breakpoint line number, the corresponding variable will be null (NULL).		R/-/-
#24101			R/-/-
#24102			R/-/-
#24103			R/-/-

B.3.6 MLC Exchange Variable

B.3.6.1 MLC M Relay Write to # Variable (#25000~#25127)

B

	+0	+1	+2	+3	+4	Type
#25000	M2X128	M2X129	M2X130	M2X131	M2X132	R/-/-
#25005	M2X133	M2X134	M2X135	M2X136	M2X137	R/-/-
#25010	M2X138	M2X139	M2X140	M2X141	M2X142	R/-/-
#25015	M2X143	M2X144	M2X145	M2X146	M2X147	R/-/-
#25020	M2X148	M2X149	M2X150	M2X151	M2X152	R/-/-
#25025	M2X153	M2X154	M2X155	M2X156	M2X157	R/-/-
#25030	M2X158	M2X159	M2X160	M2X161	M2X162	R/-/-
#25035	M2X163	M2X164	M2X165	M2X166	M2X167	R/-/-
#25040	M2X168	M2X169	M2X170	M2X171	M2X172	R/-/-
#25045	M2X173	M2X174	M2X175	M2X176	M2X177	R/-/-
#25050	M2X178	M2X179	M2X180	M2X181	M2X182	R/-/-
#25055	M2X183	M2X184	M2X185	M2X186	M2X187	R/-/-
#25060	M2X188	M2X189	M2X190	M2X191	M2X192	R/-/-
#25065	M2X193	M2X194	M2X195	M2X196	M2X197	R/-/-
#25070	M2X198	M2X199	M2X200	M2X201	M2X202	R/-/-
#25075	M2X203	M2X204	M2X205	M2X206	M2X207	R/-/-
#25080	M2X208	M2X209	M2X210	M2X211	M2X212	R/-/-
#25085	M2X213	M2X214	M2X215	M2X216	M2X217	R/-/-
#25090	M2X218	M2X219	M2X220	M2X221	M2X222	R/-/-
#25095	M2X223	M2X224	M2X225	M2X226	M2X227	R/-/-
#25100	M2X228	M2X229	M2X230	M2X231	M2X232	R/-/-
#25105	M2X233	M2X234	M2X235	M2X236	M2X237	R/-/-
#25110	M2X238	M2X239	M2X240	M2X241	M2X242	R/-/-
#25115	M2X243	M2X244	M2X245	M2X246	M2X247	R/-/-
#25120	M2X248	M2X249	M2X250	M2X251	M2X252	R/-/-
#25125	M2X253	M2X254	M2X255	-	-	R/-/-

B.3.6.2 MLC Word D Register Write to # Variable (#25128~#25255)

This function effected by the parameter **[N1.010 Bit7 MLC variable status]**. When the parameter set as 0 single word type, the D register uses 16-bit signed integer is converted into the # variable, for example: D2X128 as -19, the #25128 will read out of -19.000; if set parameter as 1 floating-point type, D register uses 32-bit floating-point format conversion passed in the # variable and odd-numbered of # variables are forbidden to operate, for example: set the value of D2X128, D2X129 as 19.999, read #25128 out of 19.999 and the #25129 is not available to operate.

	+0	+1	+2	+3	+4	Type
D2X128	#25128	#25129note	#25130	#25131note	#25132	R/-/-
D2X133	#25133note	#25134	#25135note	#25136	#25137note	R/-/-
D2X138	#25138	#25139note	#25140	#25141note	#25142	R/-/-
D2X143	#25143note	#25144	#25145note	#25146	#25147note	R/-/-
D2X148	#25148	#25149note	#25150	#25151note	#25152	R/-/-
D2X153	#25153note	#25154	#25155note	#25156	#25157note	R/-/-
D2X158	#25158	#25159note	#25160	#25161note	#25162	R/-/-
D2X163	#25163note	#25164	#25165note	#25166	#25167note	R/-/-
D2X168	#25168	#25169note	#25170	#25171note	#25172	R/-/-
D2X173	#25173note	#25174	#25175note	#25176	#25177note	R/-/-
D2X178	#25178	#25179note	#25180	#25181note	#25182	R/-/-
D2X183	#25183note	#25184	#25185note	#25186	#25187note	R/-/-
D2X188	#25188	#25189note	#25190	#25191note	#25192	R/-/-
D2X193	#25193note	#25194	#25195note	#25196	#25197note	R/-/-
D2X198	#25198	#25199note	#25200	#25201note	#25202	R/-/-
D2X203	#25203note	#25204	#25205note	#25206	#25207note	R/-/-
D2X208	#25208	#25209note	#25210	#25211note	#25212	R/-/-
D2X213	#25213note	#25214	#25215note	#25216	#25217note	R/-/-
D2X218	#25218	#25219note	#25220	#25221note	#25222	R/-/-
D2X223	#25223note	#25224	#25225note	#25226	#25227note	R/-/-
D2X228	#25228	#25229note	#25230	#25231note	#25232	R/-/-
D2X233	#25233note	#25234	#25235note	#25236	#25237note	R/-/-
D2X238	#25238	#25239note	#25240	#25241note	#25242	R/-/-
D2X243	#25243note	#25244	#25245note	#25246	#25247note	R/-/-
D2X248	#25248	#25249note	#25250	#25251note	#25252	R/-/-
D2X253	#25253	#25254	#25255	-	-	R/-/-

Note: parameter **[N1.010 Bit7 MLC variable status]** set as 1 floating-point type, the odd # variable number are prohibited.

B

B.3.6.3 # Variable Write to MLC M Relay (#25256~#25383)

	+0	+1	+2	+3	+4	Type
#25256	M2X256	M2X257	M2X258	M2X259	M2X260	R/W/-
#25261	M2X261	M2X262	M2X263	M2X264	M2X265	R/W/-
#25266	M2X266	M2X267	M2X268	M2X269	M2X270	R/W/-
#25271	M2X271	M2X272	M2X273	M2X274	M2X275	R/W/-
#25276	M2X276	M2X277	M2X278	M2X279	M2X280	R/W/-
#25281	M2X281	M2X282	M2X283	M2X284	M2X285	R/W/-
#25286	M2X286	M2X287	M2X288	M2X289	M2X290	R/W/-
#25291	M2X291	M2X292	M2X293	M2X294	M2X295	R/W/-
#25296	M2X296	M2X297	M2X298	M2X299	M2X300	R/W/-
#25301	M2X301	M2X302	M2X303	M2X304	M2X305	R/W/-
#25306	M2X306	M2X307	M2X308	M2X309	M2X310	R/W/-
#25311	M2X311	M2X312	M2X313	M2X314	M2X315	R/W/-
#25316	M2X316	M2X317	M2X318	M2X319	M2X320	R/W/-
#25321	M2X321	M2X322	M2X323	M2X324	M2X325	R/W/-
#25326	M2X326	M2X327	M2X328	M2X329	M2X330	R/W/-
#25331	M2X331	M2X332	M2X333	M2X334	M2X335	R/W/-
#25336	M2X336	M2X337	M2X338	M2X339	M2X340	R/W/-
#25341	M2X341	M2X342	M2X343	M2X344	M2X345	R/W/-
#25346	M2X346	M2X347	M2X348	M2X349	M2X350	R/W/-
#25351	M2X351	M2X352	M2X353	M2X354	M2X355	R/W/-
#25356	M2X356	M2X357	M2X358	M2X359	M2X360	R/W/-
#25361	M2X361	M2X362	M2X363	M2X364	M2X365	R/W/-
#25366	M2X366	M2X367	M2X368	M2X369	M2X370	R/W/-
#25371	M2X371	M2X372	M2X373	M2X374	M2X375	R/W/-
#25376	M2X376	M2X377	M2X378	M2X379	M2X380	R/W/-
#25381	M2X381	M2X382	M2X383	-	-	R/W/-

B.3.6.4 # Variable Write to MLC D Register (#25384~#25511)

This function effected by the parameter **[N1.010 Bit7 MLC variable status]**, when set parameter as 0 single word type, the # variable uses 16-bit signed integer conversion is passed to the D register, for example: set #25384 as -19.999, read D3X128 out of -19; if set parameter as 1 floating-point type, # variable uses 32-bit floating-point format in D register and odd-numbered of # variables are forbidden to operate, for example: set the value of #25328 as 19.999, read D3X128 and D3X129 out of 19.999 and the #25329 is not available to operate.

	+0	+1	+2	+3	+4	Type
D3X128	#25384	#25385 note	#25386	#25387 note	#25388	R/W/-
D3X133	#25389 note	#25390	#25391 note	#25392	#25393 note	R/W/-
D3X138	#25394	#25395 note	#25396	#25397 note	#25398	R/W/-
D3X143	#25399 note	#25400	#25401 note	#25402	#25403 note	R/W/-
D3X148	#25404	#25405 note	#25406	#25407 note	#25408	R/W/-
D3X153	#25409 note	#25410	#25411 note	#25412	#25413 note	R/W/-
D3X158	#25414	#25415 note	#25416	#25417 note	#25418	R/W/-
D3X163	#25419 note	#25420	#25421 note	#25422	#25423 note	R/W/-
D3X168	#25424	#25425 note	#25426	#25427 note	#25428	R/W/-
D3X173	#25429 note	#25430	#25431 note	#25432	#25433 note	R/W/-
D3X178	#25434	#25435 note	#25436	#25437 note	#25438	R/W/-
D3X183	#25439 note	#25440	#25441 note	#25442	#25443 note	R/W/-
D3X188	#25444	#25445 note	#25446	#25447 note	#25448	R/W/-
D3X193	#25449 note	#25450	#25451 note	#25452	#25453 note	R/W/-
D3X198	#25454	#25455 note	#25456	#25457 note	#25458	R/W/-
D3X203	#25459 note	#25460	#25461 note	#25462	#25463 note	R/W/-
D3X208	#25464	#25465 note	#25466	#25467 note	#25468	R/W/-
D3X213	#25469 note	#25470	#25471 note	#25472	#25473 note	R/W/-
D3X218	#25474	#25475 note	#25476	#25477 note	#25478	R/W/-
D3X223	#25479 note	#25480	#25481 note	#25482	#25483 note	R/W/-
D3X228	#25484	#25485 note	#25486	#25487 note	#25488	R/W/-
D3X233	#25489 note	#25490	#25491 note	#25492	#25493 note	R/W/-
D3X238	#25494	#25495 note	#25496	#25497 note	#25498	R/W/-
D3X243	#25499 note	#25500	#25501 note	#25502	#25503 note	R/W/-
D3X248	#25504	#25505 note	#25506	#25507 note	#25508	R/W/-
D3X253	#25509 note	#25510	#25511 note	-	-	R/W/-

Note: parameter **[N1.010 Bit7 MLC variable status]** set as 1 floating-point type, the odd # variable number are prohibited.

B.3.7 Tool Management

B

B.3.7.1 Tool Length (#26000~#41999)

	T1	T2	T3	T4	~	T1000	Type
Axis 1	#26000	#26001	#26002	#26003	~	#26999	R/W/-
Axis 2	#27000	#27001	#27002	#27003	~	#27999	R/W/-
Axis 3	#28000	#28001	#28002	#28003	~	#28999	R/W/-
Axis 4	#29000	#29001	#29002	#29003	~	#29999	R/W/-
Axis 5	#30000	#30001	#30002	#30003	~	#30999	R/W/-
Axis 6	#31000	#31001	#31002	#31003	~	#31999	R/W/-
Axis 7	#32000	#32001	#32002	#32003	~	#32999	R/W/-
Axis 8	#33000	#33001	#33002	#33003	~	#33999	R/W/-
Axis 9	#34000	#34001	#34002	#34003	~	#34999	R/W/-
Axis 10	#35000	#35001	#35002	#35003	~	#35999	R/W/-
Axis 11	#36000	#36001	#36002	#36003	~	#36999	R/W/-
Axis 12	#37000	#37001	#37002	#37003	~	#37999	R/W/-
Axis 13	#38000	#38001	#38002	#38003	~	#38999	R/W/-
Axis 14	#39000	#39001	#39002	#39003	~	#39999	R/W/-
Axis 15	#40000	#40001	#40002	#40003	~	#40999	R/W/-
Axis 16	#41000	#41001	#41002	#41003	~	#41999	R/W/-

B.3.7.2 Tool Wear (#42000~#57999)

	T1	T2	T3	T4	~	T1000	Type
Axis 1	#42000	#42001	#42002	#42003	~	#42999	R/W/-
Axis 2	#43000	#43001	#43002	#43003	~	#43999	R/W/-
Axis 3	#44000	#44001	#44002	#44003	~	#44999	R/W/-
Axis 4	#45000	#45001	#45002	#45003	~	#45999	R/W/-
Axis 5	#46000	#46001	#46002	#46003	~	#46999	R/W/-
Axis 6	#47000	#47001	#47002	#47003	~	#47999	R/W/-
Axis 7	#48000	#48001	#48002	#48003	~	#48999	R/W/-
Axis 8	#49000	#49001	#49002	#49003	~	#49999	R/W/-
Axis 9	#50000	#50001	#50002	#50003	~	#50999	R/W/-
Axis 10	#51000	#51001	#51002	#51003	~	#51999	R/W/-
Axis 11	#52000	#52001	#52002	#52003	~	#52999	R/W/-
Axis 12	#53000	#53001	#53002	#53003	~	#53999	R/W/-
Axis 13	#54000	#54001	#54002	#54003	~	#54999	R/W/-
Axis 14	#55000	#55001	#55002	#55003	~	#55999	R/W/-
Axis 15	#56000	#56001	#56002	#56003	~	#56999	R/W/-
Axis 16	#57000	#57001	#57002	#57003	~	#57999	R/W/-

B.3.7.3 Tool Radius/ Status (#42000~#57999)

	T1	T2	T3	T4	~	T1000	Type
Tool radius	#58000	#58001	#58002	#58003	~	#58999	R/W/-
Radius wear	#59000	#59001	#59002	#59003	~	#59999	R/W/-
Tool status	#60000	#60001	#60002	#60003	~	#60999	R/W/-

B.3.7.4 Tool Tolerance (#61000~#76999)

Tool tolerance # variable can be defined in the DOPSoft's tool table and then can be applied to tool wear, end tool detection by macro program.

	T1	T2	T3	T4	~	T1000	Type
Axis 1	#61000	#61001	#61002	#61003	~	#61999	R/W/-
Axis 2	#62000	#62001	#62002	#62003	~	#62999	R/W/-
Axis 3	#63000	#63001	#63002	#63003	~	#63999	R/W/-
Axis 4	#64000	#64001	#64002	#64003	~	#64999	R/W/-
Axis 5	#65000	#65001	#65002	#65003	~	#65999	R/W/-
Axis 6	#66000	#66001	#66002	#66003	~	#66999	R/W/-
Axis 7	#67000	#67001	#67002	#67003	~	#67999	R/W/-
Axis 8	#68000	#68001	#68002	#68003	~	#68999	R/W/-
Axis 9	#69000	#69001	#69002	#69003	~	#69999	R/W/-
Axis 10	#70000	#70001	#70002	#70003	~	#70999	R/W/-
Axis 11	#71000	#71001	#71002	#71003	~	#71999	R/W/-
Axis 12	#72000	#72001	#72002	#72003	~	#72999	R/W/-
Axis 13	#73000	#73001	#73002	#73003	~	#73999	R/W/-
Axis 14	#74000	#74001	#74002	#74003	~	#74999	R/W/-
Axis 15	#75000	#75001	#75002	#75003	~	#75999	R/W/-
Axis 16	#76000	#76001	#76002	#76003	~	#76999	R/W/-

B.3.7.5 Tool Lifetime Target (#77000~92999)

These variables are tool lifetime setting for multi head machine.

Lathe and milling machine, please refer to tool lifetime description (#192000~#195999)

	T1	T2	T3	T4	~	T1000	Type
Axis 1	#77000	#77001	#77002	#77003	~	#77999	R/W/-
Axis 2	#78000	#78001	#78002	#78003	~	#78999	R/W/-
Axis 3	#79000	#79001	#79002	#79003	~	#79999	R/W/-
Axis 4	#80000	#80001	#80002	#80003	~	#80999	R/W/-
Axis 5	#81000	#81001	#81002	#81003	~	#81999	R/W/-
Axis 6	#82000	#82001	#82002	#82003	~	#82999	R/W/-
Axis 7	#83000	#83001	#83002	#83003	~	#83999	R/W/-
Axis 8	#84000	#84001	#84002	#84003	~	#84999	R/W/-
Axis 9	#85000	#85001	#85002	#85003	~	#85999	R/W/-
Axis 10	#86000	#86001	#86002	#86003	~	#86999	R/W/-
Axis 11	#87000	#87001	#87002	#87003	~	#87999	R/W/-
Axis 12	#88000	#88001	#88002	#88003	~	#88999	R/W/-
Axis 13	#89000	#89001	#89002	#89003	~	#89999	R/W/-
Axis 14	#90000	#90001	#90002	#90003	~	#90999	R/W/-
Axis 15	#91000	#91001	#91002	#91003	~	#91999	R/W/-
Axis 16	#92000	#92001	#92002	#92003	~	#92999	R/W/-

B.3.7.6 Tool Lifetime Accumulation (#93000~108999)

These variables are tool actual lifetime setting for multi head machine.

Lathe and milling machine, please refer to tool lifetime description (#192000~#195999)

	T1	T2	T3	T4	~	T1000	Type
Axis 1	#93000	#93001	#93002	#93003	~	#93999	R/W/-
Axis 2	#94000	#94001	#94002	#94003	~	#94999	R/W/-
Axis 3	#95000	#95001	#95002	#95003	~	#95999	R/W/-
Axis 4	#96000	#96001	#96002	#96003	~	#96999	R/W/-
Axis 5	#97000	#97001	#97002	#97003	~	#97999	R/W/-
Axis 6	#98000	#98001	#98002	#98003	~	#98999	R/W/-
Axis 7	#99000	#99001	#99002	#99003	~	#99999	R/W/-
Axis 8	#100000	#100001	#100002	#100003	~	#100999	R/W/-
Axis 9	#101000	#101001	#101002	#101003	~	#101999	R/W/-
Axis 10	#102000	#102001	#102002	#102003	~	#102999	R/W/-
Axis 11	#103000	#103001	#103002	#103003	~	#103999	R/W/-
Axis 12	#104000	#104001	#104002	#104003	~	#104999	R/W/-
Axis 13	#105000	#105001	#105002	#105003	~	#105999	R/W/-
Axis 14	#106000	#106001	#106002	#106003	~	#106999	R/W/-
Axis 15	#107000	#107001	#107002	#107003	~	#107999	R/W/-
Axis 16	#108000	#108001	#108002	#108003	~	#108999	R/W/-

B.3.7.7 Tool Lifetime (#192000~195999)

	T1	T2	T3	T4	~	T1000	Type
Tool target lifetime	#192000	#192001	#192002	#192003	~	#192999	R/W/-
Tool actual use time	#193000	#193001	#193002	#193003	~	#193999	R/W/-
Tool target lifetime count	#194000	#194001	#194002	#194003	~	#194999	R/W/-
Tool actual used count	#195000	#195001	#195002	#195003	~	#195999	R/W/●

Smarter. Greener. Together.

Industrial Automation Headquarters

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

Asia

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

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

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

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

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

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

Australia: Delta Electronics (Australia) Pty Ltd.
Unit 2, Building A, 18-24 Ricketts Road,
Mount Waverley, Victoria 3149 Australia
Mail: IA.au@deltaww.com
TEL: +61-1300-335-823 / +61-3-9543-3720

Americas

USA: Delta Electronics (Americas) Ltd.
5101 Davis Drive, Research Triangle Park, NC 27709, U.S.A.
TEL: +1-919-767-3813 / FAX: +1-919-767-3969

Brazil: Delta Electronics Brazil Ltd.
Estrada Velha Rio-São Paulo, 5300 Eugênio de
Melo - São José dos Campos CEP: 12247-004 - SP - Brazil
TEL: +55-12-3932-2300 / FAX: +55-12-3932-237

Mexico: Delta Electronics International Mexico S.A. de C.V.
Gustavo Baz No. 309 Edificio E PB 103
Colonia La Loma, CP 54060
Tlalnepantla, Estado de México
TEL: +52-55-3603-9200

EMEA

EMEA Headquarters: Delta Electronics (Netherlands) B.V.
Sales: Sales.IA.EMEA@deltaww.com
Marketing: Marketing.IA.EMEA@deltaww.com
Technical Support: iatechnicalsupport@deltaww.com
Customer Support: Customer-Support@deltaww.com
Service: Service.IA.emea@deltaww.com
TEL: +31(0)40 800 3900

BENELUX: Delta Electronics (Netherlands) B.V.
Automotive Campus 260, 5708 JZ Helmond, The Netherlands
Mail: Sales.IA.Benelux@deltaww.com
TEL: +31(0)40 800 3900

DACH: Delta Electronics (Netherlands) B.V.
Coesterweg 45,D-59494 Soest,Germany
Mail: Sales.IA.DACH@deltaww.com
TEL: +49 2921 987 238

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

Iberia: Delta Electronics Solutions (Spain) S.L.U
Ctra. De Villaverde a Vallecas, 265 1º Dcha Ed.
Hormigueras – P.I. de Vallecas 28031 Madrid
TEL: +34(0)91 223 74 20

Carrer Llacuna 166, 08018 Barcelona, Spain
Mail: Sales.IA.Iberia@deltaww.com

Italy: Delta Electronics (Italy) S.r.l.
Via Meda 2–22060 Novedrate(CO)
Piazza Grazioli 18 00186 Roma Italy
Mail: Sales.IA.Italy@deltaww.com
TEL: +39 039 8900365

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

MEA: Eltek Dubai (Eltek MEA DMCC)
OFFICE 2504, 25th Floor, Saba Tower 1,
Jumeirah Lakes Towers, Dubai, UAE
Mail: Sales.IA.MEA@deltaww.com
TEL: +971(0)4 2690148