

Original Instructions

PARAMETER LIST ALARM LIST M-CODE LIST

HORIZONTAL CENTER NEXUS 6000-II
HORIZONTAL CENTER NEXUS 6800-II
HORIZONTAL CENTER NEXUS 8800-II
HORIZONTAL CENTER NEXUS 10800-II

Manual No. : H272HA0033E

NC: MAZATROL MATRIX NEXUS 2

Serial No. :

Before using this machine and equipment, fully understand the contents of this manual to ensure proper operation. Should any questions arise, please ask the nearest Technical Center or Technology Center.

IMPORTANT NOTICE

1. Be sure to observe the safety precautions described in this manual and the contents of the safety plates on the machine and equipment. Failure may cause serious personal injury or material damage. Please replace any missing safety plates as soon as possible.
2. No modifications are to be performed that will affect operation safety.
3. For the purpose of explaining the operation of the machine and equipment, some illustrations may not include safety features such as covers, doors, etc. Before operation, make sure all such items are in place.
4. This manual was considered complete and accurate at the time of publication, however, due to our desire to constantly improve the quality and specification of all our products, it is subject to change or modification. If you have any questions, please contact the nearest Technical Center or Technology Center.
5. Always keep this manual near the machinery for immediate use.
6. If a new manual is required, please order from the nearest Technical Center or Technology Center with the manual No. or the machine name, serial No. and manual name.

Issued by Manual Publication Section, YAMAZAKI MAZAK CORPORATION, Japan

02.2015

Serial No. 275288

Copyright © 2013 YAMAZAKI MAZAK CORPORATION. All rights reserved.
Serial No. 275288

SAFETY PRECAUTIONS

Preface

Safety precautions relating to the CNC unit (in the remainder of this manual, referred to simply as the NC unit) that is provided in this machine are explained below. Not only the persons who create programs, but also those who operate the machine must thoroughly understand the contents of this manual to ensure safe operation of the machine.

Read all these safety precautions, even if your NC model does not have the corresponding functions or optional units and a part of the precautions do not apply.

Rule

1. This section contains the precautions to be observed as to the working methods and states usually expected. Of course, however, unexpected operations and/or unexpected working states may take place at the user site.
During daily operation of the machine, therefore, the user must pay extra careful attention to its own working safety as well as to observe the precautions described below.
2. Although this manual contains as great an amount of information as it can, since it is not rare for the user to perform the operations that overstep the manufacturer-assumed ones, not all of "what the user cannot perform" or "what the user must not perform" can be fully covered in this manual with all such operations taken into consideration beforehand.
It is to be understood, therefore, that functions not clearly written as "executable" are "inexecutable" functions.
3. The meanings of our safety precautions to DANGER, WARNING, and CAUTION are as follows:



DANGER

: Failure to follow these instructions could result in loss of life.



WARNING

: Failure to observe these instructions could result in serious harm to a human life or body.



CAUTION

: Failure to observe these instructions could result in minor injuries or serious machine damage.

Basics



WARNING

- After turning power on, keep hands away from the keys, buttons, or switches of the operating panel until an initial display has been made.
- Before proceeding to the next operations, fully check that correct data has been entered and/or set. If the operator performs operations without being aware of data errors, unexpected operation of the machine will result.
- Before machining workpieces, perform operational tests and make sure that the machine operates correctly. No workpieces must be machined without confirmation of normal operation. Closely check the accuracy of programs by executing override, single-block, and other functions or by operating the machine at no load. Also, fully utilize tool path check, Virtual Machining, and other functions, if provided.
- Make sure that the appropriate feed rate and rotational speed are designated for the particular machining requirements. Always understand that since the maximum usable feed rate and rotational speed are determined by the specifications of the tool to be used, those of the workpiece to be machined, and various other factors, actual capabilities differ from the machine specifications listed in this manual. If an inappropriate feed rate or rotational speed is designated, the workpiece or the tool may abruptly move out from the machine.
- Before executing correction functions, fully check that the direction and amount of correction are correct. Unexpected operation of the machine will result if a correction function is executed without its thorough understanding.
- Parameters are set to the optimum standard machining conditions prior to shipping of the machine from the factory. In principle, these settings should not be modified. If it becomes absolutely necessary to modify the settings, perform modifications only after thoroughly understanding the functions of the corresponding parameters. Modifications usually affect any program. Unexpected operation of the machine will result if the settings are modified without a thorough understanding.

Remarks on the Cutting Conditions Recommended by the NC



WARNING

- Before using the following cutting conditions:
 - Cutting conditions that are the result of the MAZATROL Automatic Cutting Conditions Determination Function
 - Cutting conditions suggested by the Machining Navigation Function
 - Cutting conditions for tools that are suggested to be used by the Machining Navigation Function
- Confirm that every necessary precaution in regards to safe machine setup has been taken – especially for workpiece fixturing/clamping and tool setup.
- Confirm that the machine door is securely closed before starting machining. Failure to confirm safe machine setup may result in serious injury or death.

Programming



WARNING

- Fully check that the settings of the coordinate systems are correct. Even if the designated program data is correct, errors in the system settings may cause the machine to operate in unexpected places and the workpiece to abruptly move out from the machine in the event of contact with the tool.
- During surface velocity hold control, as the current workpiece coordinates of the surface velocity hold control axes approach zeroes, the spindle speed increases significantly. For the lathe, the workpiece may even come off if the chucking force decreases. Safety speed limits must therefore be observed when designating spindle speeds.
- Even after inch/metric system selection, the units of the programs, tool information, or parameters that have been registered until that time are not converted. Fully check these data units before operating the machine. If the machine is operated without checks being performed, even existing correct programs may cause the machine to operate differently from the way it did before.
- If a program is executed that includes the absolute data commands and relative data commands taken in the reverse of their original meaning, totally unexpected operation of the machine will result. Recheck the command scheme before executing programs.
- If an incorrect plane selection command is issued for a machine action such as arc interpolation or fixed-cycle machining, the tool may collide with the workpiece or part of the machine since the motions of the control axes assumed and those of actual ones will be interchanged. (This precaution applies only to NC units provided with EIA/ISO functions.)
- The mirror image, if made valid, changes subsequent machine actions significantly. Use the mirror image function only after thoroughly understanding the above. (This precaution applies only to NC units provided with EIA/ISO functions.)
- If machine coordinate system commands or reference position returning commands are issued with a correction function remaining made valid, correction may become invalid temporarily. If this is not thoroughly understood, the machine may appear as if it would operate against the expectations of the operator. Execute the above commands only after making the corresponding correction function invalid. (This precaution applies only to NC units provided with EIA/ISO functions.)
- The barrier function performs interference checks based on designated tool data. Enter the tool information that matches the tools to be actually used. Otherwise, the barrier function will not work correctly.
- The system of G-code and M-code commands differs, especially for turning, between the machines of INTEGREX e-Series and the other turning machines.
Issuance of the wrong G-code or M-code command results in totally non-intended machine operation. Thoroughly understand the system of G-code and M-code commands before using this system.

Sample program	Machines of INTEGREX e-Series	Turning machines
S1000M3	The milling spindle rotates at 1000 min ⁻¹ .	The turning spindle rotates at 1000 min ⁻¹ .
S1000M203	The turning spindle rotates at 1000 min ⁻¹ .	The milling spindle rotates at 1000 min ⁻¹ .

**WARNING**

- For the machines of INTEGREX e-Series, programmed coordinates can be rotated using an index unit of the MAZATROL program and a G68 command (coordinate rotate command) of the EIA/ISO program. However, for example, when the B-axis is rotated through 180 degrees around the Y-axis to implement machining with the turning spindle No. 2, the plus side of the X-axis in the programmed coordinate system faces downward and if the program is created ignoring this fact, the resulting movement of the tool to unexpected positions may incite collisions.
To create the program with the plus side of the X-axis oriented in an upward direction, use the mirror function of the WPC shift unit or the mirror imaging function of G-code command (G50.1, G51.1).
- After modifying the tool data specified in the program, be sure to perform the tool path check function, the Virtual Machining function, and other functions, and confirm that the program operates properly. The modification of tool data may cause even a field-proven machining program to change in operational status.

If the user operates the machine without being aware of any changes in program status, interference with the workpiece could arise from unexpected operation.

For example, if the cutting edge of the tool during the start of automatic operation is present inside the clearance-including blank (unmachined workpiece) specified in the common unit of the MAZATROL program, care is required since the tool will directly move from that position to the approach point because of no obstructions being judged to be present on this path.

For this reason, before starting automatic operation, make sure that the cutting edge of the tool during the start of automatic operation is present outside the clearance-including workpiece specified in the common unit of the MAZATROL program.

**CAUTION**

- If axis-by-axis independent positioning is selected and simultaneously rapid feed selected for each axis, movements to the ending point will not usually become linear. Before using these functions, therefore, make sure that no obstructions are present on the path.
- Before starting the machining operation, be sure to confirm all contents of the program obtained by conversion. Imperfections in the program could lead to machine damage and operator injury.

Operations

**WARNING**

- Single-block, feed hold, and override functions can be made invalid using system variables #3003 and #3004. Execution of this means the important modification that makes the corresponding operations invalid. Before using these variables, therefore, give thorough notification to related persons. Also, the operator must check the settings of the system variables before starting the above operations.
- If manual intervention during automatic operation, machine locking, the mirror image function, or other functions are executed, the workpiece coordinate systems will usually be shifted. When making machine restart after manual intervention, machine locking, the mirror image function, or other functions, consider the resulting amounts of shift and take the appropriate measures. If operation is restarted without any appropriate measures being taken, collision with the tool or workpiece may occur.
- Use the dry run function to check the machine for normal operation at no load. Since the feed rate at this time becomes a dry run rate different from the program-designated feed rate, the axes may move at a feed rate higher than the programmed value.
- After operation has been stopped temporarily and insertion, deletion, updating, or other commands executed for the active program, unexpected operation of the machine may result if that program is restarted. No such commands should, in principle, be issued for the active program.

**CAUTION**

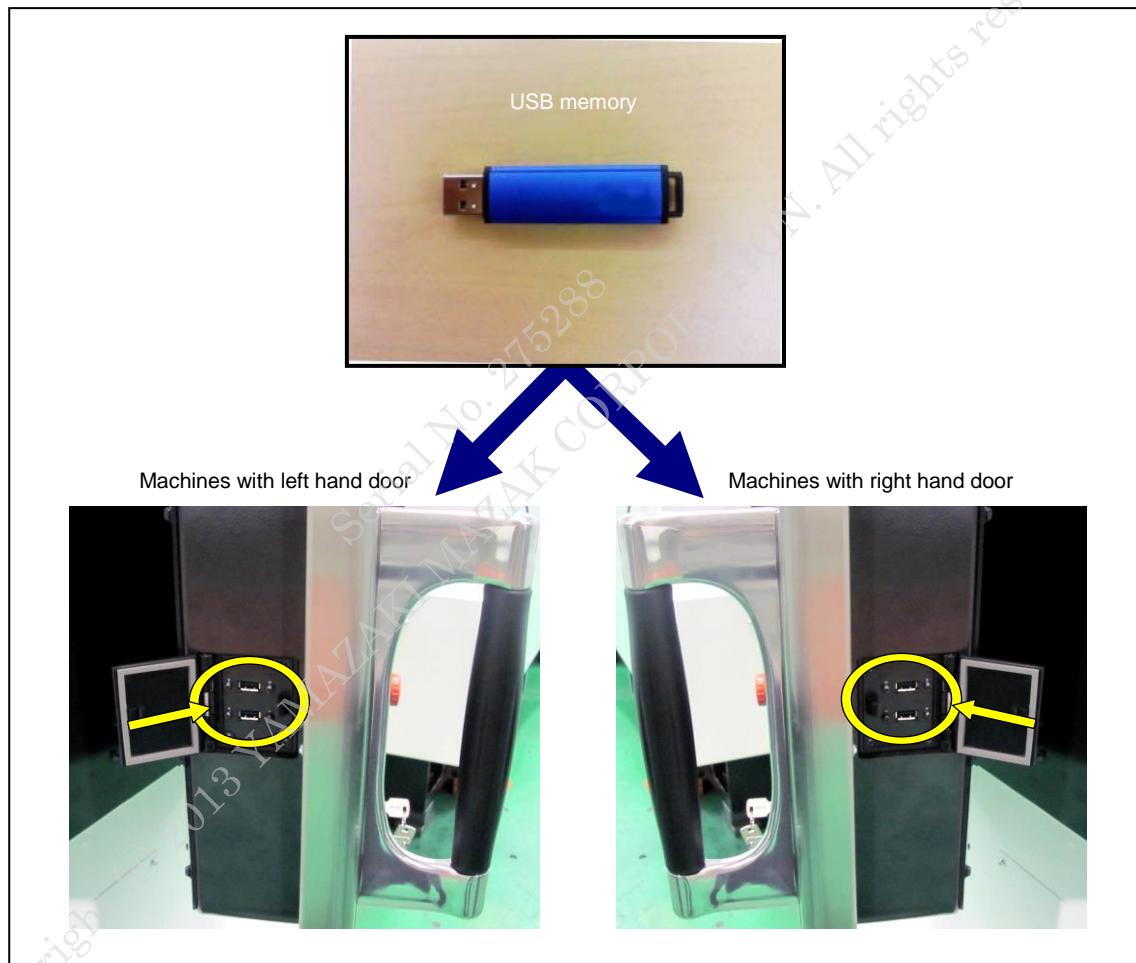
- During manual operation, fully check the directions and speeds of axial movement.
- For a machine that requires manual homing, perform manual homing operations after turning power on. Since the software-controlled stroke limits will remain ineffective until manual homing is completed, the machine will not stop even if it oversteps the limit area. As a result, serious machine damage will result.
- Do not designate an incorrect pulse multiplier when performing manual pulse handle feed operations. If the multiplier is set to 1000 times and the handle operated inadvertently, axial movement will become faster than that expected.

REQUEST TO THE USER

Request for Saving Data of Machining Programs

Machining programs saved on the hard disk of the NC unit may not be read out in the event of accidental hard disk trouble. The user, therefore, is earnestly requested to back up and store every machining program of importance at regular intervals onto an external memory (USB memory, memory card, etc.).

- The procedure for data storage is detailed in the Operating Manual, Part 3 (OPERATING NC UNIT AND PREPARATION FOR AUTOMATIC OPERATION), Chapter 9, (DISPLAY RELATED TO DATA STORAGE).
- Always use an initialized USB memory. The location of the USB connector depends on the machine model, as shown below.



On machines with a random ATC feature, each actual ATC operation changes the tool data (in pocket numbers). Be sure not to run the machine after loading the externally stored data of the **TOOL DATA** display without having confirmed the data's correspondence to the current tooling on the magazine. Otherwise the machine cannot be guaranteed to operate normally.

BEFORE USING THE NC UNIT

Limited Warranty

The warranty of the manufacturer does not cover any trouble arising if the NC unit is used for its non-intended purpose. Take notice of this when operating the unit.

Examples of the trouble arising if the NC unit is used for its non-intended purpose are listed below.

1. Trouble associated with and caused by the use of any commercially available software products (including user-created ones)
2. Trouble associated with and caused by the use of any Windows operating systems
3. Trouble associated with and caused by the use of any commercially available computer equipment

Operating Environment

1. Ambient temperature

During machine operation: 0° to 50°C (32° to 122°F)

2. Relative humidity

During machine operation: 10 to 75% (without bedewing)

Note : As humidity increases, insulation deteriorates causing electrical component parts to deteriorate quickly.

Keeping the Backup Data

Note : Do not attempt to delete or modify the data stored in the following folder.

Recovery Data Storage Folder: D:\MazakBackUp

Although this folder is not used when the NC unit is running normally, it contains important data that enables the prompt recovery of the machine if it fails.

If this data has been deleted or modified, the NC unit may require a long recovery time. Be sure not to modify or delete this data.

BEFORE USING THE NC UNIT



Copyright (c) 2013 YAMAZAKI MAZAK CORPORATION. All rights reserved.
Serial No. 275288

CONTENTS

1	INTRODUCTION	1-1
2	PARAMETER	2-1
2-1	Outline	2-1
2-1-1	Types of parameters	2-1
2-1-2	Precautions	2-2
2-2	Parameter List	2-3
2-2-1	User parameter	2-3
2-2-2	Machine parameter	2-33
2-2-3	Data I/O parameter	2-63
2-3	Detailed Description	2-67
2-3-1	Structure of the parameter list	2-67
2-3-2	User parameter POINT (D)	2-69
2-3-3	User parameter LINE/FACE/3D (E)	2-94
2-3-4	User parameter EIA/ISO (F)	2-118
2-3-5	User parameter SOFT LIMIT (I)	2-190
2-3-6	User parameter SYSTEM (SU)	2-195
2-3-7	User parameter TURNING (TC)	2-216
2-3-8	User parameter SOLID (SD)	2-257
2-3-9	Machine parameter CALL MACRO (J)	2-262
2-3-10	Machine parameter MEASURE (K)	2-270
2-3-11	Machine parameter TABLE (L)	2-311
2-3-12	Machine parameter FEED VEL. (M)	2-351
2-3-13	Machine parameter TIME CONST. (N)	2-367

2-3-14	Machine parameter ANOTHER (S)	2-374
2-3-15	Machine parameter SPINDLE (SA).....	2-383
2-3-16	Machine parameter BARRIER (BA)	2-403
2-3-17	Data I/O parameter CMT parameter (CMT)	2-432
2-3-18	Data I/O parameter TAPE parameter (TAP)	2-435
2-3-19	Data I/O parameter DNC parameter (DNC)	2-446
2-3-20	Data I/O parameter OTHER (IOP/DPR/IDD).....	2-455
3	ALARM	3-1
3-1	Outline	3-2
3-1-1	Alarm display	3-2
3-1-2	Precautions.....	3-3
3-2	Detailed Description	3-4
3-2-1	Structure of the alarm list.....	3-4
3-2-2	No. 1 - No. 99, No. 1000 - No. 1099 (System/Drive error).....	3-5
3-2-3	No. 100 - No. 199, No. 1100 - No. 1199 (CNC machine control error).....	3-12
3-2-4	No. 200 - No. 399, No. 1200 - No. 1399 (PLC machine control error).....	3-27
3-2-5	No. 400 - No. 499, No. 1400 - No. 1499 (CNC screen operation error).....	3-41
3-2-6	No. 500 - No. 599, No. 1500 - No. 1599 (I/O error)	3-61
3-2-7	No. 600 - No. 699, No. 1600 - No. 1699 (MAZATROL program error)	3-76
3-2-8	No. 700 - No. 799, No. 1700 - No. 1799 (MAZATROL program error)	3-92
3-2-9	No. 800 - No. 899, No. 1800 - No. 1899 (EIA/ISO program error).....	3-104
3-2-10	No. 900 - No. 999, No. 1900 - No. 1999 (EIA/ISO program error).....	3-121
3-2-11	No. 2000 - No. 2099 (3D setup error)	3-134
3-2-12	No. 2100 - No. 2199 (Interference error)	3-138

3-2-13	No. 2200 - No. 2299 (CAM data output error)	3-141
4	M-CODE LIST.....	4-1

Copyright (c) 2013 YAMAZAKI MAZAK CORPORATION. All rights reserved.
Serial No. 275288



Copyright(c) 2013 YAMAZAKI MAZAK CORPORATION. All rights reserved.
Serial No. 275288

1 INTRODUCTION

This manual describes the meaning and setting of various parameters, and the meaning and elimination procedure of various alarms used for the MAZATROL MATRIX System. This document also gives the list of M-codes.

For detailed description of the MAZATROL MATRIX System, refer to the Operating Manual of the machine.

Read this manual and the Operating Manual of the machine carefully in order to make the best use of the possibilities of the MAZATROL MATRIX System.

Copyright (c) 2013 YAMAZAKI MAZAK CORPORATION. All rights reserved.
Serial No. 275288

1

INTRODUCTION



Copyright(c) 2013 YAMAZAKI MAZAK CORPORATION. All rights reserved.
Serial No. 275288

2 PARAMETER

2-1 Outline

1. Scope of this chapter

This chapter describes the parameters you can change as required. How to read the list is described in the beginning. Always refer to this list to change parameters.

2. Precautions on this chapter

This chapter also gives parameters relating to optional functions. Accordingly, the list includes parameters which cannot be changed. Check the type of machine purchased by you and its specifications before you read the list.

Note 1: The contents of this list are subjected to change without notice, for NC unit or machine improvement.

Note 2: Any questions about the contents of this list should be communicated to Mazak Technical Center or Technology Center.

2-1-1 Types of parameters

Parameters, which refer to constants specific to the NC machines and equipment and the data necessary for cutting operations, possess a very important meaning.

Parameters can be broadly divided into the following three types according to their meaning.

- User parameters

The data required for processes such as point machining, line machining, plane machining, turning, and EIA/ISO programmed machining, is registered. The **USER PARAMETER** display is used to register the user parameters.

- Machine parameters

Constants related to the servomotors and spindle motors, machine status data etc. are registered. The **MACHINE PARAMETER** display is used to register the machine parameters.

- Data I/O parameters

The data required for connection to external units such as a CMT unit and a tape unit, is registered. The **DATA I/O PARAMETER** display which can be selected on the **DATA I/O** display is used to register the data I/O parameters.

2-1-2 Precautions

1. Details of the parameters may differ according to the machine used, the presence/absence of an option(s), the production time of the NC machines and equipment, etc. Therefore, do not use the parameters of other machines.
2. The parameter list is supplied in the form of data sheets within the NC electronic cabinet at shipment of the machines. Be careful not to lose the list.
3. Before making changes to details of a parameter, make sure that the parameter is the one to be changed.
4. If details of the parameter to be changed cannot be clearly understood, contact Mazak Technical Center or Technology Center.
5. When changing details of a parameter, maintain records of the old and new data.
6. If the particular machine is not used for a long time, then the battery to protect the parameter memory will run down. (Battery alarm)
In that case, errors will occur in the parameters and thus machine malfunctions may result.
To prevent this, first check the existing details of the parameters closely against the separate parameter list and then make the necessary changes to the parameters.
7. In addition to the parameters listed in this document, those related to PLC (Programmable Logic Controller) are also available; refer to the OPERATING MANUAL of the machine for details of the PLC-related parameters and the PLC Parameter List in the ELECTRIC WIRING DIAGRAM.

2-2 Parameter List

2-2-1 User parameter

1. POINT (D)

Address (bit)	Outline
D1	Height of the second R-point during point machining
D2	Nominal diameter of spot-machining tool
D3	Number of revolutions during dwell at hole bottom in spot-machining cycle
D4	Maximum allowable spot-chamfering hole diameter element
D5	Prehole through speed during inversed spot-facing
D6	Drill-machining cycle setting element
D7	Drill-machining cycle setting element
D8	Maximum diameter of holes machinable on one drill
D9	Maximum diameter of holes machinable on two drills
D10	Maximum diameter of holes machinable on three drills
D11	Through-hole/tap-prehole machining overshoot
D12	Stop-hole machining hole-bottom clearance
D13	Spot-machining hole diameter (fixed value)
D14	Depth-of-cut setting element for drilling (ALMINUM)
D15	Depth-of-cut setting element for drilling (except AL)
D16	Number of revolutions during dwell at hole bottom for chamfering cutter or spot-machining tool in chamfering cycle
D17	Interference clearance of chamfering cutter
D18	Return feed rate for reaming or boring (cycle 3)
D19	Number of revolutions during dwell at hole bottom for end milling
D20	Radial depth-of-cut setting element for end milling
D21	Reference bottom-finishing allowance for end milling
D22	Tapping-cycle dwell time
D23	Prehole clearance for end milling

Address (bit)	Outline
D24	Number of revolutions during dwell at hole bottom for boring
D25	Boring-bar tip relief
D26	Returning distance from hole bottom for boring or back-boring
D28	Bottom-finishing amount of boring
D29	Chip removal time
D30	Number of incomplete threads in tapping cycle
D31	Tapper elongation amount for tapping
D32	Number of spindle revolutions until spindle CCW rotation begins in tapping cycle
D33	Back-boring tool tip relief
D35	Prehole-drilling diameter setting element for reamer (drilling)
D36	Prehole-drilling diameter setting element for reamer (boring)
D37	Prehole-drilling diameter setting element for reamer (end milling)
D38	Reamer-prehole diameter setting element for boring or end milling
D39	Reamer-prehole diameter setting element for end milling
D40	Number of revolutions during dwell at spot-faced hole bottom for inversed spot-facing
D41	R-point height during point-machining
D42	Height of the third R-point during point machining
D43	Number of incomplete threads in tapping cycle for piped screw
D44	Selecting an automatic calculation method for the amount of chamfer with the tapping unit
D45	Gradual decrements in drilling depth
D46	Minimum gradual drilling depth
D47	Reamer-prehole machining overshoot
D48	Feed override for the section to be chamfered in the planetary tapping cycle
D49	Amount of return at hole bottom during the planetary tapping cycle
D50	Auto-set feed rate for pre-hole machining in the planetary tapping cycle
D51	Auto-set feed rate for planetary tapping cycle
D52	Reduction ratio for the G00-based relief rate during a very-deep-hole drilling cycle

Address (bit)	Outline
D53	Number of times of pecking up to the return of the tool to a position near the starting point of the very-deep-hole drilling cycle of a drilling or turning-drilling unit
D54	Deceleration rate at cutting start for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle
D55	Drilling return distance for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle
D56	Number of revolutions during dwell at chip ejection position and hole bottom for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle
D57	Return speed for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle
D58	Feed rate reduction distance ratio at cutting start of a very-deep-hole drilling cycle/decremental very-deep-hole machining cycle
D59	Surface speed reduction ratio at cutting end of a very-deep-hole drilling cycle/decremental very-deep-hole machining cycle
D60	Automatic setting ratio of axial cutting feed rate during chamfering
D61	Nominal diameter of chamfering cutter, set during automatic tool development for a point machining unit
D73 - D77	Learning of cutting conditions (DEP-Z range)
D78 - D82	Learning of cutting conditions (WID-R range)
D91 0	M04 is output/not output after the tool has dwelled at the hole bottom during a tapping cycle.
D91 1	The tool dwells/does not dwell after M04 has been output at the hole bottom during a tapping cycle.
D91 2	The tool dwells/does not dwell after it has been returned to the R-point during a tapping cycle.
D91 3	If a drill is used in the pre-machining of the centering drill cycle, the R-point height is set to D1 or not.
D91 4	The finishing tool path is shortened/not shortened during a true-circle processing cycle (end milling).
D91 5	The tool path is shortened/not shortened during a true-circle processing cycle (chamfering).
D91 6	If a pre-machining tool sequence is included in the same unit, the R-point height of the drill is set/not set to D1 or D42.
D91 7	The R-point height of the chamfering cutter during the cycle 2 is set to D42 or not. The R-point height of the spot-machining tool during the chamfering cycle (cycle 2) is set to D42 or not. R-point height selection for smooth chamfering (point machining)
D92 0	During a true-circle processing (end milling) cycle, E17 is used for axial feed or not.
D92 1	The R1-point height of the back spot facing is set to D1 or not.
D92 2	If a chamfering cutter is included in the premachining tool sequence of the same unit, the R-point height of the reamer is set to D1 or not.
D92 3	If a chamfering cutter is included in the pre-machining tool sequence of the same unit, the R-point height of the tapping is set to D1 or not.
D92 4	The position cleared radially or axially by D17 from the workpiece section concerned is subjected to check for interference or not.
D92 5	Dwell time designation for synchronous tapping in a MAZATROL program valid/invalid
D92 6	During planetary tapping, chips are ejected/not ejected automatically prior to the threading process.
D93	Unidirectional positioning for point-machining

Address (bit)	Outline
D94	Unidirectional positioning for point-machining
D95	Auto-setting method for tapping
D96 0	Surface speed calculation for spot drill used
D96 1	Element that determines the counterboring pattern
D96 3	Menu display setting of RGH BCB (inversed faced hole machining), BK-CBORE (back boring) and CBOR-TAP (counterbore tapping), as well as BACKSPOT FACER (inversed spot-facing tool) and BACK BORING BAR (back boring) in the tool sequence data
D97 0	Cutting drawing method for circular milling unit on the VIRTUAL MACHINING display
D105 0	Handling the X-axial coordinates of non-turning TPC relay points
D105 1	Tap high-speed deep-hole cycle menu display/hidden
D106 0	Machine operation when machining units are not preceded by an indexing unit
D106 1	Execution of an index command after the second or subsequent work coordinate command in WPC unit
D106 2	T-code output during indexing before the end unit
D106 4	Selection of rotational axis command system by table selection
D106 5	Alarm output timing when the first machining unit is not preceded by an index unit

2. LINE/FACE/3D (E)

Address (bit)	Outline
E1	Closed-pattern cutting start point and escape point setting element
E2	Cutting start point and escape point setting element (the first clearance)
E4	Reference allowance of finishing in radial direction
E5	Element used to set the cutting start point and escape point (the second clearance)
E6	Reference allowance of finishing in axial direction
E7	Allowance of cutting start point in axial direction (the second clearance)
E8	Radial interference clearance of chamfering cutter
E9	Allowance of axial-cutting start position (the first clearance)
E10	Depth-of-cut-R automatic setting element (Face milling, End milling-top, End milling-step)
E11	Axial interference clearance of chamfering cutter
E12	Radial interference clearance of face milling unit and angular face milling unit

Address (bit)	Outline
E13	Tool path setting element for end milling-top unit
E14	Depth-of-cut-R automatic setting element (Pocket milling, Pocket milling-mountain, Pocket milling-valley)
E15	Tool path setting element for face milling-top unit (reciprocating short)
E16	Peripheral-cutting feed rate override for end milling-mountain unit
E17	Axial-cutting feed rate override
E18	Override in case of the overall width cutting for pocket-machining
E19	Returning feed rate override in case of bidirectional cutting for rough-machining of the end milling-slot unit.
E20	Axial cutting feed override during Z-axial cutting in the pecking mode of face machining
E21	Wall-cutting overlap in closed figure
E22	Override value of automatic corner over-riding
E23	Effective removal allowance (upper limit) of automatic corner overriding
E24	Effective removal allowance (lower limit) of automatic corner overriding
E25	Effective angle (upper limit) of automatic corner overriding
E26	Calculation coefficient for the finishing feed of line milling
E27	Radial direction feed rate calculation reference diameter for finish cutting in line machining unit
E28	Finishing feed rate calculation reference feed rate in line machining unit
E29	Selection of whether the cutting conditions in the shape sequence during VFC mode are to be modified
E30	An element that determines the starting point and escape point of radial cutting when CLOSED is specified for the wall attributes at the starting point and ending point of open-pattern line machining
E31	Element that determines the amount of OPEN attribute wall protrusion in pocket-machining shape units
E32	Element that automatically determines an approaching radius in a Z-direction helical approach scheme
E33	Approaching gradient for a Z-direction helical approach scheme
E34	Element that automatically determines an approaching distance in a Z-direction tapered approach scheme
E35	Approaching gradient for a Z-direction tapered approach scheme
E36	Element that automatically determines an escape distance in the Z-direction tapered escape scheme
E37	The amount of return of pecking in the Z-axial pecking mode of face machining
E38	The returning feed rate of pecking in the Z-axial pecking mode of face machining
E55	3-D, Axial cutting-feed overriding
E56	3-D, Inversion check of curved-surface pattern

Address (bit)	Outline
E57	3-D, Severity check of cutting pitch
E58	3-D, Tool-diameter compensation
E59	3-D, Allowance of axial-cutting start position
E60	3-D, Normal cutting allowance
E61	3-D, Search length for parallel cutting
E62	3-D, Search length for right-angle cutting
E63	3-D, Pattern display division segment (FL direction)
E64	3-D, Pattern display division segment (GL direction)
E65	3-D, Radial cutting allowance for area check
E66	3-D, Axial cutting allowance for area check
E67 - E75	3-D, Processing error tolerance
E76	3-D, Entire-width override
E77	3-D, Radial cutting allowance for high-speed rough processing (workpiece size appointment)
E78	3-D, Multiplying factor set for tolerance
E83	3-D, Region of radial machining during high-speed rough processing (offset appointment)
E84	3-D, Region of axial machining during high-speed rough processing (offset appointment)
E85	3-D, Region of radial machining during high-speed rough processing: -X (workpiece size appointment)
E86	3-D, Region of radial machining during high-speed rough processing: +X (workpiece size appointment)
E87	3-D, Region of radial machining during high-speed rough processing: -Y (workpiece size appointment)
E88	3-D, Region of radial machining during high-speed rough processing: +Y (workpiece size appointment)
E89	3-D, Region of axial machining during high-speed rough processing (workpiece size appointment)
E91	Tool-path pattern selection for end milling-mountain unit
E92	Tool-path pattern selection for pocket milling unit
E93	Tool-path pattern selection for pocket milling-mountain unit
E94	Tool-path pattern selection for pocket milling-valley unit
E95	Tool-path pattern selection for line-machining unit
E96	Tool-path pattern selection for end milling-slot unit
E97	Tool-path pattern selection for end milling-top unit

Address (bit)	Outline
E98	Cutting method selection for end milling-mountain, pocket milling-valley unit
E99 0	Milling feed rate specification range for the shape sequence of the MAZATROL program
E99 1	Bottom/wall simultaneous finishing mode for pocket machining
E99 3	Interference check between the approach point and the cutting starting point of face machining
E99 4	Alarm generating condition during the approach or escape for line machining
E104 0	Cutting method to be used after an approach point is automatically determined during line/face machining
E104 1	Returning position during face machining
E104 2	Returning position during line machining
E104 3	Infeed position when a CLOSED attribute is assigned to the wall of line machining
E104 6	Calculation of the ZC machining feed rate
E105 0	B-axis user-defined angle setting during C-axis central line machining
E105 1	Selection of the surface on which to describe a line machining shape in the ZC mode
E106 1	Selection of a relay point to be used before TPC escapement (line/face machining unit)

3. EIA/ISO (F)

Address (bit)	Outline
F1	G61.1 corner deceleration coefficient (%)
F2	G61.1 arc-clamping speed coefficient (%)
F3	For high-speed smoothing control
F4	Distance to judge the shape formed by 3 axes under tool tip point control in the 5-axis fairing function
F5	Distance to judge the shape formed by rotary 2 axes in the 5-axis fairing function
F6	Minimum allowable height of stepped sections for deceleration in high-speed smoothing control mode
F7	Fixed value (0)
F8	Corner deceleration speed coefficient for high-speed smoothing control
F9	Circular cutting clamp speed coefficient for high-speed smoothing control
F10	Return amount during tap high-speed deep-hole cycle
F11	Vector constant for 3-D, tool-diameter compensation
F12	Return amount of pecking in drill high-speed deep-hole cycle or in G73

Address (bit)	Outline
F13	Allowance amount of rapid-feed stop in deep-hole drilling cycle or in G83/G283
F14	Rotation center of coordinates (axis of abscissa)
F15	Rotation center of coordinates (axis of ordinate)
F16	Horizontal length of coordinate rotation
F17	Vertical length of coordinate rotation
F18	Angle of coordinate rotation
F19	Maximum permissible difference in arc radius
F20	Fixed value of scaling factor
F21	Maximum inside-corner angle available with automatic corner override (G62)
F22	Deceleration area of automatic corner overriding (G62)
F27	Handling of G92 (spindle speed clamp value) command at restart
F28	Threading chamfering angle
F29	Override value of automatic corner overriding (G62)
F30	G-code type selection
F31	Selection of a roughing priority type for program layout
F32	Operation when argument R is omitted from the maximum/minimum spindle speed setting
F33 0	Incorporation of the MAZATROL tool length into the current position counter valid/invalid
F33 1	Type of the turning tool to be automatically set in the MAZATROL program, created by loading an M640T or T-NEXUS program
F33 4	Selection of restart setting method in the MAZATROL program
F33 6	Modal indication for G-code group No. 1 in the fixed cycle mode
F33 7	Modal indication on "radius data input for X-axis" in the mode of 3D coordinate conversion (G68.5)
F34 0	Cancellation of the ram spindle offset command (G52.1) when the RESET key is pressed valid/invalid
F34 1	Representation on the POSITION display for G49 (only for the INTEGREX e Series)
F34 2	Parts count override in the middle of automatic operation
F34 4	Programming type for inclined-surface machining
F34 6	Compensating movement for a T-code with a movement command in one block
F34 7	Application of Z-OFFSET and C-OFFSET values for the tape operation mode

Address (bit)	Outline
F35 0	Selection whether or not multiple material pieces can be arranged on the VIRTUAL MACHINING or 3D MONITOR display
F35 1	Automatic setting of the material model for a MAZATROL program of initial-point scheme on the VIRTUAL MACHINING or 3D MONITOR display
F35 2	When assigning G-code for a cutting function not to be represented with the INTELLIGENT SAFETY SHIELD being selected for automatic operation, operation is continued/alarm state
F35 3	When a program prepared and stored for the Hard Disk operation is called up as a subprogram with the INTELLIGENT SAFETY SHIELD being selected for automatic operation, operation is continued/alarm state
F35 4	Representation of cutting-off or parting operation by turning on the VIRTUAL MACHINING and 3D MONITOR displays
F35 6	Representation of cutting-off or parting operation by milling on the VIRTUAL MACHINING and 3D MONITOR displays
F35 7	Alarming/No alarming when the feed-overriding value has been changed in the flow of automatic operation with the INTELLIGENT SAFETY SHIELD being active
F36 1	Automatic M-code setting in EIA/ISO program modal restarting mode
F36 2	Automatic S-code setting in EIA/ISO program modal restarting mode
F36 3	Error check function in cases where numerical data is omitted in the EIA program or in MDI mode
F36 4	Display of the chuck and tailstock areas for an EIA/ISO program on the TRACE or TOOL PATH CHECK display
F36 6	Number of effective decimal digits in the tool axis vector's components (I, J, K) for tool tip point control type 2 (G43.5)
F36 7	Rotational axis prefILTERing
F37 0	Temporary cancellation of geometry compensation for G0 (rapid traverse) in the mode of tool tip point control
F37 1	Subprogram call given in manual program unit (for turning machines only)
F37 2	Tool length compensation in MDI operation
F37 3	Timing of clearing #3006 (stop) message
F37 6	Operation when "ATC" is not specified in end unit
F38 1	Handling of tool index numbers (after tool data exchange)
F38 2	Font size setting for EIA/ISO PROGRAM display and EIA MONITOR display
F38 4	Negative tool diameter setting
F39	Plane to be used for an EIA/ISO program (called by a subprogram unit) and a manual program unit
F40	Operating method selection in tape mode
F41	Threading termination waiting time processing
F42	Deceleration area r during Z-axis measurement
F43	Measurement area d during Z-axis measurement
F44	Measuring speed f

Address (bit)	Outline
F45	Deceleration area r during X-axis measurement
F46	Deceleration area d during X-axis measurement
F47 - F66	Common variable name
F69	EIA/ISO program restart method
F70	Availability of multiple-machining and designated number of repetitions in the EIA/ISO subprogram
F71	Machining order control
F72	Selection of the shape correction function of the MAZATROL program
F73	M-code execution time for time study
F74	S-code execution time for time study
F75	T-code execution time for time study
F76	B-code execution time for time study
F77	Basis rate for tool life judgment
F78	Selection of separating ratio of graphic display
F79 0	Holding of memory monitor address valid/invalid
F79 1	Selection of menu display
F79 2	Key history function valid/invalid
F79 3	Tool search method
F79 4	Selection of tap gear
F79 5	Display of tools currently in use valid/invalid
F79 6	Initial value of synchronous/asynchronous tapping during tapping tool registration
F79 7	Display of a MAZATROL monitor window valid/invalid
F80 0	MAZATROL function valid/invalid
F80 1	Automatic display of the navigation window on the occurrence of an alarm valid/invalid
F80 2	MAINTENANCE CHECK display at power on, displayed/not displayed
F80 3	Third page of the MAINTENANCE CHECK display, displayed/not displayed
F80 4	GRAPHIC MAINTENANCE display on the occurrence of an alarm, displayed/not displayed
F80 5	Learning of cutting conditions valid/invalid

Address (bit)	Outline
F80 6	Editing on the CUTTING CONDITION LEARN display valid/invalid
F80 7	Destination of spare tool correction by the workpiece measurement
F81 0	Program editing prohibited (in 9000's)
F81 1	Program displaying prohibited (in 9000's)
F81 2	Fixed value (0)
F81 3	Add-in MAZATROL valid/invalid
F81 4	Program editing prohibited (in 8000, 9000's)
F81 5	Program displaying prohibited (in 8000, 9000's)
F81 6	Retaining the program transfer settings valid/invalid
F81 7	Input error prevention function valid/invalid
F82 0	Characteristics estimation result graph, displayed/not displayed
F82 1	Selection of inch/metric representation in POSITION display, TOOL DATA display, and TOOL OFFSET display modes valid/invalid
F82 2	Basis for tool life judgment
F82 3	—
F82 4	X-axis diameter display on the POSITION display valid/invalid
F82 5	Whether the stored tools registration function on the VISUAL TOOL MANAGEMENT display is to be made valid or invalid when the visual tool ID/data management functions are valid
F82 6	Automatic setting of the grooving tool length
F82 7	Index tool wear correction setup during automatic operation
F83 0	Output of the alarm history data as text data valid/invalid
F83 1	Operation record function valid/invalid
F83 2	—
F83 3	—
F83 4	—
F83 5	—
F83 6	—
F83 7	Attribute of the starting/ending point of line machining after loading a program of M640 series onto the MATRIX (only for a program of Z-offset scheme)
F84 0	Selection of whether or not the tool offset data is taken into account for the POSITION counter during the execution of EIA/ISO programs

Address (bit)	Outline
F84 1	Switching of arguments B and J in a fixed cycle
F84 2	Spare tool search method for EIA/ISO programs
F84 3	Timing to validate new workpiece offset data specified with a system variable
F84 4	Machine coordinate system (G92) selection
F84 5	Incremental/absolute data command in high-speed machining mode
F84 6	Tape run method
F84 7	Operation executed/not executed when the tool data has not bee set
F85 0	Table rotational machining
F85 1	Radial interference check
F85 2	Type of coordinate system for controlling the tool tip point
F85 3	Tool tip point control scheme
F85 4	Fixed value (0)
F85 5	Reset to cancel G68.2, valid/invalid
F85 6	Display of surface definition θ
F85 7	Output of the B-axis unclamping code before B-axis indexing, valid/invalid
F86 0	Output of M250 (Spindle Speed Confirmation) before a turning feed, valid/invalid
F86 1	Milling-spindle start timing for a milling unit (with MILL&TURN. set under TYPE of UNo. 0)
F86 2	Override scheme for G0 during tool tip point control
F86 4	Display of the PART SHAPE window
F86 5	Override scheme for G1 during tool tip point control
F86 6	Selection of rotary axis reference position for tool tip point control
F87 0	Selection of whether or not the machine is to be offset by each change only in the deviation vector
F87 1	Selection of whether or not the check for mismatch of the workpiece origin and table rotation center is to be conducted
F87 2	Data alteration checking function valid/invalid (tool-related function)
F87 3	Feed rate display method for the superposition system
F87 4	Single-block mode cancellation during fixed cycle invalid/valid
F87 5	Common variables (#500 and on) are cleared by resetting or at the program end, invalid/valid

Address (bit)	Outline
F87 6	Local variables are cleared by resetting or at the program end, invalid/valid
F87 7	Common variables (#100 to #199) are cleared by resetting or at the program end, invalid/valid
F88	Specifications related to the conversion from MAZATROL program into an EIA program
F89 0	[For EIA conversion] Output of shape data
F89 1	[For EIA conversion] G-codes for synchronous tapping: G74/G84 or G84.2/G84.3
F89 2	[For EIA conversion] Output of F command
F89 3	[For EIA conversion] Mill-turning unit conversion, alarm/shifted path
F89 4	[For EIA conversion] Output destination, standard area/ backup area
F89 5	—
F89 6	[For EIA conversion] Returning to the second zero point before tool change (output of G30)
F89 7	Fixed value (0)
F91 0	In response to move command without decimal point, tool moves by 1/tool moves by 10
F91 1	Coordinate system shift using a MAZATROL program, valid/invalid
F91 2	Stroke inside check before movement/Stroke outside check before movement
F91 3	—
F91 4	Metric (Initial G20 is valid/invalid)/Inch
F91 5	In response to move command without decimal point:
F91 6	G00 interpolation/non-interpolation
F91 7	G33E command is for the number of threads per inch/command is for thread cutting with precise lead
F92 0	Modal at power-on or at reset (G17 or G19/G18)
F92 1	Modal at power-on or at reset (G17 or G19/G18)
F92 2	Dwell command method
F92 3	Tool-length compensation (G43, G44) axis for milling tools
F92 4	Tool-diameter compensation (G41 or G42) start up/cancel type
F92 5	Tool-diameter compensation (G41 or G42) interference check
F92 6	Fixed-cycle hole-drilling axis
F92 7	ACT-Φ/NOSE-R in the TOOL DATA display for an EIA/ISO program, invalid/valid

Address (bit)	Outline
F93 0	—
F93 1	Modal at power-on or at reset (G94/G95)
F93 2	Modal at power-on or at reset (G91/G90)
F93 3	Tool length of tool data for an EIA/ISO program, invalid/valid
F93 4	Feed rate during machine lock
F93 5	Middle point during reference-point return
F93 6	Single-block operation mode at user macro operation instruction
F93 7	Fixed value (0)
F94 0	Movement to hole-drilling position in fixed-cycle mode
F94 1	External deceleration signal valid/invalid
F94 2	Tool length offsetting during G28/G30 execution, canceled/Performed
F94 3	Modal at power-on or at reset (G01/G00)
F94 4	Tool command method using T-codes
F94 5	Fixed value (0)
F94 6	Fixed value (1)
F94 7	Tool offset amount effectuated in an EIA/ISO program
F95 0	Interrupt function using user macro instruction, valid/invalid
F95 1	Handling of macroprogram interruption and call
F95 2	Automatic return position to restart the program (Fixed to 1)
F95 3	G00 (positioning) command feed rate for dry run
F95 4	—
F95 5	Alarm display when the slave or adaptor unit is not connected
F95 6	Manual-pulse interrupt amount cancellation with reset key, valid/invalid
F95 7	Coordinate system corresponding to G54 set with reset key, valid/invalid
F96 0	Selection of variable number for tool offset amount
F96 1	Fairing function valid/invalid
F96 2	Processing for arc command blocks in high-speed machining mode, nonuniform feed/uniform feed

Address (bit)	Outline
F96 3	—
F96 4	Selection of a corner judgment criterion in high-speed machining mode
F96 5	Selection of a cutting feed clamping speed in high-speed machining mode
F96 6	Rotational axis shape correction valid/invalid
F96 7	—
F97	Selection of G-code of the coordinates system to be used in the EIA conversion function
F98	Number of macro variable to be used in the EIA conversion function
F99	Offset amount for the subprogram WNo. to the main WNo. concerned in case of output with subprogram in the EIA conversion function
F100	Spline cancel length
F101	Spline cancel angle
F102	Fine spline interpolation curve error (Block including the point of inflection)
F103	Spline interpolation fairing block length, 5-axis spline interpolation small block length
F104	Fine spline interpolation curve error (block including no inflection point), 5-axis spline interpolation curve error, 5-axis fairing command error
F107	Small block judgment length
F108	Corner deceleration angle increment value
F109	Number of common variables between turrets (#100 onward)
F110	Number of common variables between turrets (#500 onward)
F111 0	Selection of display type of tapping tool on the VIRTUAL MACHINING or 3D MONITOR display
F111 1	Use/disuse of dry run during thread cutting
F111 2	Use/disuse of feed hold during thread cutting
F111 3	Direction of rotation of the C-axis during C-axial threading with G01.1
F111 5	Tool correction amount selection for EIA/ISO programs
F111 6	Execution mode selection for a fixed turning cycle
F111 7	Form of single-block stop during a fixed turning cycle
F112	Selection of measurement data items to be printed out
F113 0	Counting all types of use under the same tool number for the tool life management on the TOOL DATA display executed/not executed
F113 1	Data handling on the milling tool of a group that has expired in tool life

Address (bit)	Outline
F113 2	Data handling on the turning tool of a group that has expired in tool life
F113 3	Tool life management of the FLASH tool
F113 4	Tool life management — Life time
F113 5	Tool life management — Maximum available wear offset data X
F113 6	Tool life management — Maximum available wear offset data Y
F113 7	Tool life management — Maximum available wear offset data Z
F114 0	Selection of the maximum C-axial cutting feed rate for the inch system
F114 1	Selection of the operation occurring during the control of the tool tip point when command G49 is issued (when the tool length offset value is canceled)
F114 2	B-axis angle check during measurement of the turning tool for which OUT is specified in "section to be machined", valid/invalid
F114 3	Moving axes by using G49 (tool length cancel) in G43 (tool length offset) mode, valid/invalid
F114 4	Selecting a rethreading function
F114 5	Output timing of a tool life alarm
F114 6	Initial setting of G53.5
F114 7	Judging tool life from the count of machined workpieces, valid/invalid
F115	Restart/TPS approach speed
F116	Feed rate of the threading runout — X-axis
F117	Feed rate of the threading runout — Y-axis
F118	Feed rate of the threading runout — Z-axis
F119	Runout feed rate for the inside diameter threading cycle
F120	Clamping speed for the threading cycle — X-axis
F121	Clamping speed for the threading cycle — Y-axis
F122	Clamping speed for the threading cycle — Z-axis
F124	Permissible data alteration amount 1 for input error prevention function
F125	Permissible data alteration amount 2 for input error prevention function
F133	Pitch of tapping tool for display in detail on the VIRTUAL MACHINING or 3D MONITOR display
F134	Thread depth of tapping tool for display in detail on the VIRTUAL MACHINING or 3D MONITOR display
F135	Tool-drawing accuracy on the VIRTUAL MACHINING or 3D MONITOR display

Address (bit)	Outline
F136	Amount of offset for dummy workpiece shape on the VIRTUAL MACHINING or 3D MONITOR display
F137	Number of jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for No. 1 turning spindle
F138	Number of jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for No. 2 turning spindle
F139	Angle offset for the jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for the No. 1 turning spindle
F140	Angle offset for the jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for the No. 2 turning spindle
F141	Roughness level of the circular milling unit (precision tornado cycle)
F143 0	Operation when workpiece counting is made valid while the automatic operation INTELLIGENT SAFETY SHIELD is enabled
F143 1	Status of the [STORE WORKPCE MATERIAL] menu function when the power is turned on
F143 2	Operation when a specific G-code is issued while the automatic operation INTELLIGENT SAFETY SHIELD is enabled
F143 6	Display of WPC during inclined-surface machining on the POSITION display
F143 7	G43 modal cancel selection with the reset key
F144 1	Selection of table rotary axis reference position for inclined-surface machining
F144 3	Setting a minus value for argument Q in hobbing, valid/invalid
F144 5	Selection of operation by the G49 command immediately after a macro variable command
F144 6	Deletion of unnecessary codes during EIA/ISO conversion for machining centers
F144 7	Deletion of G92.5 blocks during EIA/ISO conversion for machining centers
F145	Rapid feed override when data alteration is detected
F146	Selection of the material model's part to be taken off (hidden) on the VIRTUAL MACHINING and 3D MONITOR displays in the flow of turning operation under synchronization of both spindles
F147	Selection of the material model's part to be taken off (hidden) on the VIRTUAL MACHINING and 3D MONITOR displays in the flow of milling operation
F148	Compensation amount for G41/G42 given in a manual program (MANL PRG) unit
F149	Operation between blocks when single block mode is switched from ON to OFF during automatic operation
F150	Output data offset of NC basic information for field network
F151 7	Time stamp used at the time of read/write command to macro variable #3001 (power-on time)
F152	Fixed value (0)
F153	SMART font display, valid/invalid
F154	Parameter for system internal setting Setting prohibited
F155 3	Operation when argument I is omitted from the tap cycle command having argument Q

Address (bit)	Outline
F156 2	Separation between rotary axis and linear axis when moving axes during workpiece placement error correction
F156 3	Selection of solution to workpiece placement error correction (primary rotary axis sign prioritized)
F156 4	Pitch division in the case when the arc length exceeds the maximum value during helical/spiral interpolation
F156 5	Single-block stop when the VFC key is pressed with override set to 100%
F156 6	Fixed value (0)
F157 2	EIA tip position memory
F157 5	Add-in EIA function
F158 0	Tool length/tool diameter check function after tool change
F158 1	Output timing of MSTB command after a movement command under geometry compensation
F159 0	Timing of data creation for the MACHINING NAVIGATION - RESULT display with workpiece counting made valid
F159 1	Modal selection at power on or at reset (G64/G61.1)
F159 2	Fixed value
F159 3	Fixed value
F159 4	Whether or not to display on the POSITION display a spindle output diagram stored in the setting file
F160 7	Priority change when erasing the tool path from the TOOL PATH CHECK display or TRACE display
F161 0	Shape/wear offset number separation, valid/invalid
F161 1	Shape offset handling
F161 2	Tool offset timing
F161 3	Tool offset vector handling if reset function is executed
F161 4	Shape offset handling if offset number 0 is entered
F161 5	Simplified wear offset, valid/invalid
F161 6	C/Z offset when a MAZATROL program is called from an EIA program
F161 7	C/Z offset when an EIA program is called from a MAZATROL program
F162 0	Movement/No movement according to the particular amount of offset during independent start of tool tip point control
F162 1	Type of passage of tool tip point through singular point
F162 2	Chamfer/corner R-command address selection
F162 3	Fixed hole-machining cycle return selection

Address (bit)	Outline
F162 4	6 digits in T-command for turning
F162 5	Use of the M Pro scheme as the method of selecting the Length correction axis bit
F162 6	MAZATROL program check for missing Z-offset, valid/invalid
F162 7	Encoder polarity selection
F163 0	Bar feeder scheduling function, valid/invalid
F163 1	Incorporation of wear offset data into the current position display in EIA/ISO program mode, valid/invalid
F163 2	Incorporation of wear offset data into the current position display in MAZATROL program mode, valid/invalid
F163 3	Position of thread turning tool nose on the VIRTUAL MACHINING display
F163 4	Barrier check on the VIRTUAL MACHINING display valid/invalid
F163 5	Menu on the DATA I/O display (floppy disk), displayed/hidden
F163 6	Menu on the DATA I/O display (tape), displayed/hidden
F163 7	Menu on the DATA I/O display (CMT), displayed/hidden
F164	Automatic tool data setting conditions
F165 0	High-speed synchronous tapping function, valid/invalid
F165 1	X-axis movement to minus side during polar coordinate interpolation, enabled/disabled
F165 2	Polar coordinate interpolation of the C-axis, in radius/diameter
F165 3	C-axis indexing when EIA subprogram is called from MAZATROL program
F165 4	Modal or non-modal state of Q command in deep hole drilling cycle
F165 5	Conversion of tool set data for milling tool based on head swivel angle when G53.5 is commanded
F165 6	Behavior of automatic operation of an EIA program when Z-offset is not set
F165 7	Setting at CONTI. of the end unit during tool path check, valid/invalid
F166 0	Alteration of tool set value (tool length) on the TOOL DATA display in the automatic operation mode, enabled/disabled
F166 1	Type of wear offset indicated in the milling tool list on the TOOL DATA display
F166 2	ID No./Tool name selection on the TOOL DATA display
F166 3	Selection whether the selected plane on the TRACE or TOOL PATH CHECK display is to be held
F166 4	Operation conducted when the [RESTART] menu key is pressed following completion of the search in EIA modal restarting mode
F166 5	Tool diameter auto-setting for the end mill, face mill, and ball end mill on the TOOL DATA display

Address (bit)	Outline
F166 6	Display of the "INTELLIGENT SAFETY SHIELD status specification menu" in the manual operation mode
F166 7	Automatic backup function
F167 0	Threading (by turning) shift angle (Q-command) unit
F167 1	Handling the diameter/radius of point R during the hole-machining fixed cycle
F167 2	Handling the tool measured using the G137H command
F167 3	Handling the single-process mode when operation is completed/reset
F167 4	Connecting the path linearly/Alarm, if the arc internal correction value is greater than the arc radius
F167 5	Handling the X-axis command in G68.5
F167 6	Wear correction direction
F167 7	Updating the current position by a press of the reset key in manual operation mode
F168 0	Selection of operation to perform when dynamic offset II is canceled
F168 2	Rotational center offset is added/not added to the tool length when the tool axial tool length offset command (G43.1) is issued.
F168 3	Axis for the milling tool wear correction and TOOL EYE compensation amount for the B-axis rotational turret
F168 4	Conversion of 3D coordinate conversion into inclined-surface machining
F168 5	Holding the amount of manual handle pulse interruption
F168 6	[RESTART2NONMODAL] menu function during stop of an EIA program execution invalid/valid
F168 7	Correction pattern for the groove/protrusion width measurement

4. SOFT LIMIT (I)

Address (bit)	Outline
I1	Shift amount of unidirectional positioning (G60)
I2	Upper (plus direction) user soft-limit
I3	Lower (minus direction) user soft-limit
I5	Function for making the G0 speed variable, Variable override: Minimum value
I7	Function for making the G0 speed variable, Variable control area (lower limit side)
I8	Function for making the G0 speed variable, Variable control area (upper limit side)
I9	Function for making the G0 speed variable, Variable control area lower limit

Address (bit)	Outline
I10	Function for making the G0 speed variable, Variable control area upper limit
I11	Rotary center of a workpiece
I12	Clamping value for the amount of handle interruption
I13 0	Execution of G28 (reference-point return)
I13 1	Manual zero-point return operation
I13 2	—
I13 3	—
I13 4	—
I13 5	—
I13 6	Removal of control axes, valid/invalid
I13 7	—
I14 0	Mirror image with respect to the machine zero point, valid/invalid
I14 1	—
I14 2	User software limits (I2, I3) valid/invalid
I14 3	Tool-tip relief after spindle orientation during execution of G75, G76, G86 or point-machining (boring or back-boring), valid/invalid
I14 4	Direction of tool-tip relief after spindle orientation during execution of G75, G76, G86 or point-machining (boring or back-boring)
I14 5	—
I14 6	—
I14 7	—
I17	Amount of work hand rotational error correction (Work hand angle: 0°)
I18	Amount of work hand rotational error correction (Work hand angle: 45°)
I19	Amount of work hand rotational error correction (Work hand angle: 90°)
I21	Fixed point 1 specified by the user
I22	Fixed point 2 specified by the user

5. SYSTEM (SU)

Address (bit)	Outline
SU1	Reference axis of abscissa for plane selection
SU2	Axis 1 parallel to the axis of abscissa for plane selection
SU3	Axis 2 parallel to the axis of abscissa for plane selection
SU4	Reference axis of ordinate for plane selection
SU5	Axis 1 parallel to the axis of ordinate for plane selection
SU6	Axis 2 parallel to the axis of ordinate for plane selection
SU7	Reference height axis for plane selection
SU8	Axis 1 parallel to the height axis for plane selection
SU9	Axis 2 parallel to the height axis for plane selection
SU10	Selection of tool change command output position
SU11	Movement of axes during approach
SU12	Rotating position specified in the indexing unit after tool change
SU13	Axis name of the transfer axis
SU14	Tool nose mark display color on the TOOL PATH CHECK display/ TRACE display
SU15	Name of thrust axis for servo axis
SU16	Movement to C-axis index swivel position when Z-offset scheme is used
SU17	Background color for the MDI window
SU18	Character color for the MDI window
SU19	Selecting the method of axis movement to the tool change command output position or to the position designated at RETURN of the end unit
SU20	Coefficient for polygonal machining spindle acceleration
SU21	Selection of returning operation after the execution of the workpiece transfer unit corresponding to the work hand angle to be changed
SU49	Delay timer for the parts catcher
SU50	Tool turning clearance (radial value) in X-axis
SU51	Tool turning clearance in Z-axis
SU52	Lower-turret retraction function - Tool number of the retraction tool 1
SU53	Lower-turret retraction function - Tool number of the retraction tool 2

Address (bit)	Outline
SU97 - SU100	Lower-turret retraction function - Fixed point of the retraction position
SU101	Return distance (radial value) in X-axis at wall during rough cutting in bar machining or in corner machining of EIA/ISO program
SU102	Return distance in Z-axis at wall during rough cutting in bar machining or in corner machining of EIA/ISO program
SU103	Cutting depth in the composite-type fixed cycle
SU104	Pecking return distance in grooving of EIA/ISO program
SU105	Cut depth (diametral value) for final cut in composite-type thread cutting cycle of EIA/ISO program
SU106	Minimum cut depth clamping value (radial value) in composite-type thread cutting cycle G276 of EIA/ISO program
SU107	Safety clamp speed
SU108	Safeguarding strength
SU109	Tip mass
SU110 - SU113	Tool length setting range check function - settable data range
SU123	Admissible motion range on the primary rotational axis for workpiece placement error correction
SU124	Admissible motion range on the secondary rotational axis for workpiece placement error correction
SU125	Fixed value
SU128	Retraction position of the upper and lower turrets at the end of the two-workpiece machining sequence (X-axis)
SU129	Retraction position of the upper and lower turrets at the end of the two-workpiece machining sequence (Z-axis)
SU153 0	M-code selection for tapping cycle
SU153 1	Display of BUFFER on the POSITION display
SU153 2	Display of REMAIN on the POSITION display
SU153 3	Display of POSITION on the POSITION display
SU153 4	Automatic retraction of the lower turret for MAZATROL programs
SU153 5	"C-axis Clamp" machining method for MAZATROL programs (line/linear machining for the C-axis CYLIND mode)
SU153 6	MAZATROL point machining with C-axis clamped (M210)
SU153 7	"C-axis Clamp" machining method for MAZATROL programs (line/linear machining for the C-axis FACE mode)
SU154 0	Displaying tool-setting values for ATC-type turret
SU154 1	Tool length setting range check function
SU154 2	Display of BUFFER during G18.2/G18.3 mode

Address (bit)	Outline
SU154 3	Display of REMAIN during G18.2/G18.3 mode
SU154 4	Display of POSITION during G18.2/G18.3 mode
SU154 5	Display of BUFFER during G17.1 - G17.5 mode
SU154 6	Display of REMAIN during G17.1 - G17.5 mode
SU154 7	Display of POSITION during G17.1 - G17.5 mode
SU155 0	Fixed value (1)
SU155 1	Correction timing of the touch sensor tool length during EIA/ISO programmed or MDI operation (for machining center specifications only)
SU155 2	Operation at an EIA subprogram call from a MAZATROL program
SU155 3	Judgment method for tool change command output during MAZATROL programmed operation
SU155 4	Tool change command output at the start of MAZATROL programmed operation
SU155 7	Reference value for judging tool breakage during laser tool measurement
SU156 4	Type of deep hole drilling cycle G83/G87
SU156 6	Execution of the restarting search for a block in the mode of G68 when tool length compensation (G43/G44) remains valid with a turning tool mounted
SU156 7	Execution of the restarting search for a block after the cancellation of the cross command
SU157 1	The scheme of axis movement during an approach and during movement to the tool change command output position for machining at the work hand side
SU157 2	Conditions for returning to the tool change command output position in the manual program unit (MANL PRG)
SU157 3	Movement to the tool rotating position for tool change command for the same TNo.
SU157 4	Operation to be performed if either TURN POS X, Y, or Z of the indexing unit is left blank.
SU157 5	Preparatory C-axis positioning to 0° for a manual program unit (MANL PRG) with a milling tool after a turning unit
SU157 7	Selection of whether or not the B-axis rotation radius is to be included into the "clearance position" for changing the indexed cutting edge on one and the same FLASH tool
SU158 2	Checking the rotational axes' correction values in workpiece placement error correction for MAZATROL programs
SU159 0	"TOOL NAME ORDER" display/hidden with random ATC machines

6. TURNING (TC)

Address (bit)	Outline
TC1	Cut depth reduction rate for rough cutting in bar machining unit, corner machining unit, and copy machining unit
TC2	Acceleration rate in up-going taper for rough cutting in bar machining unit
TC3	Acceleration rate in up-going wall slope (90°) for rough cutting in bar machining unit
TC4	Selection of chamfering pattern in rough external turning cycle G71 (G271) or rough edge-machining cycle G72 (G272)
TC5	Deceleration rate in down-going taper for rough cutting in bar machining unit
TC6	Deceleration rate in down-going wall slope (90°) for rough cutting in bar machining unit
TC7	Acceleration rate on outside stock contour for rough cutting in copy machining unit
TC8	Acceleration pitch error ratio in thread cutting unit
TC9	Rough cutting residue ratio in cutting off cycle in groove cutting unit
TC10	Cut depth allowable incremental rate for rough cutting in groove cutting unit, edge machining unit and copy machining unit
TC11	Deceleration rate at cutting start in turning-drilling unit
TC12	Deceleration rate at cutting end in turning-drilling unit
TC13	Deceleration rate at rough cutting start in bar machining unit and copy machining unit
TC14	Maximum permissible rate of increase of the initial cutting depth during roughing
TC15	Deceleration clearance at start of rough cutting in bar machining unit and copy machining unit
TC16	Tolerance for escape in high speed rough cutting cycle of bar machining unit
TC17	Pitch error correction during threading acceleration
TC18	Fixed value (0)
TC19	Turning-drilling cut depth calculation coefficient
TC20	Reamer return speed calculation coefficient in the turning-drilling unit
TC21	Incomplete threading portion length calculation coefficient for turning-tap tip
TC22	Turning-tapper elongation calculation coefficient
TC23	Thread height calculation coefficient for outside diameter, face/rear thread cutting (metric)
TC24	Thread height calculation coefficient for inside diameter thread cutting (metric)
TC25	Thread height calculation coefficient for outside diameter, face/rear thread cutting (inch)
TC26	Thread height calculation coefficient for inside diameter thread cutting (inch)
TC27	Recessing width for #1 to #3

Address (bit)	Outline
TC28	Recessing depth #1 to #3
TC29	Recessing width for #4
TC30	Recessing depth for #4
TC31	Recessing width for #5
TC32	Recessing depth for #5
TC33	Recessing width for #6
TC34	Recessing depth for #6
TC37	Safety contour clearance — Outside diameter clearance (radial value)
TC38	Safety contour clearance — Inside diameter clearance (radial value)
TC39	Safety contour clearance — Front clearance
TC40	Safety contour clearance — Back clearance
TC41	Thread cutting clearance (radial value)
TC42	Groove cutting clearance (radial value) in X-axis
TC43	Groove cutting clearance in Z-axis
TC44	Workpiece transfer clearance
TC45	Amount of the edge clearance after edge-machining
TC46	Drilling depth decrement in turning-drilling unit
TC47	Pecking return distance in turning-drilling unit
TC48	Drilling cut depth clamp value in turning-drilling unit
TC49	Spindle speed clamp value in cutting off cycle
TC50	Number of times that the feed rate is to be reduced during the #4 and #5 cutting-off cycles of a grooving unit
TC51	Dwell at the hole bottom during non-through hole drilling cycle of the turning-drilling unit
TC52	Dwell (specification of spindle rotation number) at groove bottom in groove cutting unit
TC53	Feed rate for escape by short distance
TC54	Cut depth per cycle for machining inside diameter in bar machining unit
TC55	Reverse feed tolerance for contour machining
TC56	Overtravelling in X-axis direction in edge machining unit
TC57	Workpiece pressing speed in workpiece transfer unit

Address (bit)	Outline
TC58	Spindle speed (min^{-1}) of two spindles in workpiece transfer while the spindles are rotating in workpiece transfer unit
TC59	Workpiece pressing distance in workpiece transfer unit
TC61	Simultaneous operation pattern for transfer
TC62	Selection of tool change command output position for FLASH tool
TC63	Amount of relief after transfer using the workpiece transfer unit (Spindle mode 0 to 5)
TC64	Amount of relief after transfer using the workpiece transfer unit (Spindle mode 6 and 7)
TC65	Specification of first M-code for parts catcher control
TC66	Minimum index angle of the FLASH tool
TC67	Return distance (radial value) in X-axis at wall during rough cutting in bar cutting unit, in corner machining unit or in mill-turning unit of MAZATROL programs
TC68	Return distance in Z-axis at wall during rough cutting in bar cutting unit, in corner machining unit or in mill-turning unit of MAZATROL programs
TC69	Number of revolutions during dwell for pecking of grooving
TC70	FLASH tool — Number of cutting edges to be used for the tool not registered in the tool file
TC71	Feed stopping rotation dwell time during the chip cutting cycle
TC72	Number of times of roughing in the composite-type fixed cycle (G273, G73)
TC73	Return speed at pecking portion in groove cutting unit and turning-drilling unit
TC74	Pecking return distance in groove cutting unit of MAZATROL programs
TC75	Overlap distance for machining wide groove in groove cutting unit
TC76	Escape value after machining in edge machining unit
TC77	Acceleration distance clamp value for thread cutting unit
TC78	Cut depth (radial value) for final cut in thread cutting unit of MAZATROL programs
TC79	Minimum cut depth clamping value (radial value) in thread cutting unit of MAZATROL programs
TC80	Angle of the tool nose during the G276/G76 mode
TC81	Final finishing repeat times in the composite-type fixed cycle (G276, G76)
TC82	Chamfering data calculation coefficient in thread cutting unit and thread cutting cycle (G276/G292, G76/G92)
TC83	Number of cutting operations to be performed on finishing allowance corresponding to standard pattern (#0) of threading unit
TC84	Feed rate to be auto-set for finishing
TC85 - TC94	Specification of the pocket for the long boring bar

Address (bit)	Outline
TC95	Fixed value
TC96	Fixed value
TC97	Type of retraction during workpiece transfer
TC98	Returning operation after machining specified in the end unit
TC99	ATC operation after machining when not specified in the end unit
TC101	Selection of droop sampling axis
TC102	Selection of cycle counter sampling axis
TC103	Amplitude limit of table vibration
TC104	Clearance for rechucking using the workpiece transfer unit
TC106	Designation of MOVE C in a workpiece transfer unit of a program of workpiece scheme (only for the INTEGREX e-H Series)
TC111 - TC113	CUTTING CONDITION LEARN display — Workpiece length range
TC114 - TC116	CUTTING CONDITION LEARN display — Max. workpiece outside diameter range
TC117	Composite-type fixed cycle — G273 amount of X-axial release
TC118	Composite-type fixed cycle — G273 amount of Z-axial release
TC120 - TC137	Distance to the front end of the long boring bar
TC141 0	Use/disuse of acceleration in up-going slope during rough cutting cycle in bar machining unit
TC141 1	Use/disuse of deceleration in down-going slope during rough cutting cycle in bar machining unit
TC141 2	Selection between use/disuse of acceleration distance check at start of thread cutting unit
TC141 3	Selection between start position shift/start angle shift for thread number offset in thread cutting unit
TC141 4	Selecting an angle margin for nose shape compensation
TC141 5	Selecting an angle margin for nose shape compensation
TC141 6	CHUCK JAW DATA display name/code selection
TC141 7	Whether to make the partition plate and the workpiece barrier valid
TC142 0	Fixed value (0)
TC142 1	Selecting an inter-unit relief path when a succession of I.D. turning units using the same tool exist and there is no movement to the rotating position of the tool
TC142 2	Selection of the jaw data reference method
TC142 3	Using angle tool holder valid/invalid

Address (bit)	Outline
TC142 5	Selection whether an alarm is to be issued if the ending position of workpiece pressing is reached during transfer of the workpiece
TC142 6	X-axis retraction position during workpiece transfer as specified in the workpiece transfer unit of the MAZATROL program
TC142 7	Upper turret retraction during machining with the lower turret (for MULTIPLEX series)
TC143	Whether the end tool of the long boring bar can be changed
TC144 0	Automatic selection of the relief path for the continuous I.D. machining
TC144 1	Movement of the workpiece transfer axis for opposed turret machine
TC144 2	C-axis clamping during workpiece transfer with C-axis positioning, valid/invalid
TC144 3	Automatic output of spindle rotation command when turning tool is used in the manual program unit
TC144 4	ATC and RETURN of the end unit after single-process operation
TC144 5	ATC and RETURN of the end unit for repetition (CONTI. = 1)
TC144 6	Relief path G00/G01 selection for edging
TC144 7	Handling when no tools are specified for the manual program unit
TC145 0	OFFSET unit Z-axial inversion during turning
TC145 1	Handling of the G04 command in the manual program unit
TC145 2	Operation by cutting-feed override 0 in G31 command mode
TC145 3	Mode selection after C-axis transfer using the workpiece transfer unit
TC145 4	Selection of a calculation method for tail barrier end reference
TC145 5	Output of maximum spindle speed command specified in the common unit
TC145 6	Alarm display when tools different in shape (infeed angle, nose angle and nose R) are designated for the upper and lower turret in balanced cutting
TC145 7	Operation of the path inversed with respect to the traveling direction of the tip (only for copy machining)
TC146 0	Selection of the turning spindle rotation timing when the relay point for the turning approach is specified
TC146 1	X-axis command in the manual program unit
TC146 2	Tool path when the infeed angle or the relief angle is 0° or less
TC146 3	C-axis command output for the second workpiece transfer unit onward when workpiece transfer units of a BAR moving pattern exist in succession
TC146 4	—
TC146 5	Automatic setting of M8 as an M-code to be outputted at the beginning of a unit which is selected for the single-process operation
TC146 6	C-offset values to be used on HD1 and HD2 for a two-workpiece machining unit

Address (bit)	Outline
TC146 7	Whether or not to modify the numeric positional indication with respect to the workpiece coordinate system according to the tool offset setting in the TPC data
TC147 0	Modal condition about diameter/radius value input for the unit succeeding a manual program or subprogram unit
TC147 1	Z/C offset value to be used in the EIA/ISO subprogram after workpiece transfer in the MAZATROL program
TC147 4	Whether pressing is to be made selectable for workpiece transfer units of a BAR moving pattern
TC147 5	Tool nose position of an inside-turning grooving tool for machining on the back side
TC147 7	Z-axial withdrawal after M-code unit execution
TC148 0	Display of an icon for threading position storage
TC149 5	Machine operation on tool path check of a program for which workpiece scheme is selected and T. CENTER is specified for X and Y , but Z is left blank in the basic coordinates system unit (WPC)

7. SOLID (SD)

Address (bit)	Outline
SD49	Machine coordinate selection
SD50	Table type
SD51	Round table axis direction
SD97	Distance of model movement per time
SD98	Amount of model rotation per time
SD125	Minimum table dimension X
SD126	Maximum table dimension X
SD127	Minimum table dimension Y
SD128	Maximum table dimension Y
SD129	Minimum table dimension Z
SD130	Maximum table dimension Z
SD131	Amount of table chamfer

2-2-2 Machine parameter

1. CALL MACRO (J)

Address (bit)	Outline
J1 - J40	G-code macroprogram call
J41 - J80	M-code macroprogram call (J44, J48, J52, J56, J60: Parameter for system internal setting Setting prohibited)
J89 - J107	Parameter for system internal setting Setting prohibited
J108	Maximum number of retrials to update the model with the SAFETY SHIELD function
J109 - J112	Barrier setup chuck reference position 1 on the No. 2 spindle side/NC tailstock position
J113 - J116	Barrier setup chuck reference position 2 on the No. 2 spindle side
J119 - J120	Barrier-setting/cancelling M-code to be used for barrier teaching
J131 - J140	Setting the M-code for making the INTELLIGENT SAFETY SHIELD function valid/invalid during automatic operation
J141 - J142	Selecting whether to output the M-code for making the INTELLIGENT SAFETY SHIELD function valid/invalid during automatic operation
J144 0	Tool interference check type
J144 2	Tool life data overwrite during automatic operation
J145 - J184	M-code macroprogram call (extended)
J265 - J424	G-code macroprogram call (extended)

2. MEASURE (K)

Address (bit)	Outline
K1	Rotational radius of the C-axis
K2	Minimum rotational angle
K3	Identification number of the shaping control axis
K7	Unbalanced axis
K10	Fixed value
K11	Selection of language to be displayed
K12	Fixed value (0)
K13	Measurement skip feed rate (X-axis, Z-axis)
K14	Measurement approach speed (X-axis, Z-axis)

Address (bit)	Outline
K15	Measurement skip speed (C-axis)
K16	Measurement approach speed (C-axis)
K17	Specification of measuring tolerance (lower limit) [valid only for L106 bit 6 = 1]
K18	Specification of measuring tolerance (upper limit) [valid only for L106 bit 6 = 1]
K19	Measurement stroke for workpiece measurement
K20	Measurement stroke for tool nose measurement
K21	Coefficient to determine rotation angle when retrying measurement C reference face
K22	Measurement retry frequency when retrying reference face C measurement
K23	Retry frequency for workpiece measurement
K24 - K28	Parameter for system internal setting Setting prohibited
K29	Simultaneous control: Delay counter for automatic correction of synchronizing errors
K30	Approach speed for laser tool length measurement
K31	Approach speed for laser tool diameter measurement
K32	Pre-measuring speed for laser tool length measurement
K33	Pre-measuring speed for laser tool diameter measurement
K34	Pre-measuring spindle speed for laser tool length measurement
K35	Pre-measuring spindle speed for laser tool diameter measurement
K36	Parameter for system internal setting Setting prohibited
K37	External deceleration speed
K38	Work number called during S-code macroprogram appointment
K39	Work number called during T-code macroprogram appointment
K40	Work number called during second auxiliary function macroprogram appointment
K41	G31 skipping speed
K42	G31.1 skipping speed
K43	G31.2 skipping speed
K44	G31.3 skipping speed
K45	G31.4 skipping speed

Address (bit)	Outline
K46	Excessive pressing error spread (Amount of drooping)
K48	Spindle FB alarm detection method
K49	First number of the standby M-codes
K50	Total number of the standby M-codes
K51	M-code during workpiece measurement retry operation
K52	Parameter for system internal setting Setting prohibited
K53	Vocal output language selection
K54	Vocal output sound level
K55	Vocal output warning reference value
K56	Name of second auxiliary function
K57	Type of S-code macroprogram appointment call
K58	Type of T-code macroprogram appointment call
K59	Type of second auxiliary function macroprogram appointment call
K60	Fixed value (4)
K61	Fixed value (1)
K62	Fixed value (1)
K63	Fixed value (1)
K64	Fixed value (2)
K65	Fixed value (1)
K66	Fixed value (1)
K67	Fixed value (1)
K68 0	Coordinate system with reference to which the offset amounts are to be used in tool length offset for the selected machining surface (For machines of five-surface machining by angular tools)
K68 4	Fixed value (0)
K68 5	Override/dryrun speed valid/invalid when the G31 command is issued
K68 7	Whether all the barrier cancel data in the program is to be cleared by executing the MMS unit in a MAZATROL program
K69 0	G31.1 skip condition SKIP-2
K69 1	G31.1 skip condition SKIP-10

Address (bit)	Outline
K69 2	G31.1 skip condition SKIP-3
K69 3	G31.1 skip condition SKIP-11
K69 4	G31.1 skip condition SKIP-5
K69 5	G31.1 skip condition SKIP-13
K69 6	G31.1 skip condition SKIP-6
K69 7	G31.1 skip condition SKIP-14
K70 0	G31.2 skip condition SKIP-2
K70 1	G31.2 skip condition SKIP-10
K70 2	G31.2 skip condition SKIP-3
K70 3	G31.2 skip condition SKIP-11
K70 4	G31.2 skip condition SKIP-5
K70 5	G31.2 skip condition SKIP-13
K70 6	G31.2 skip condition SKIP-6
K70 7	G31.2 skip condition SKIP-14
K71 0	G31.3 skip condition SKIP-2
K71 1	G31.3 skip condition SKIP-10
K71 2	G31.3 skip condition SKIP-3
K71 3	G31.3 skip condition SKIP-11
K71 4	G31.3 skip condition SKIP-5
K71 5	G31.3 skip condition SKIP-13
K71 6	G31.3 skip condition SKIP-6
K71 7	G31.3 skip condition SKIP-14
K72	G31.4 skip conditions
K73	G4 skip conditions
K74	Emergency stop contactor cutoff time (Safety supervisory function)
K75	Contactor control output device 1 (Safety supervisory function)
K76	Contactor control output device 2 (Safety supervisory function)

Address (bit)	Outline
K77	Door switch input device (Safety supervisory function)
K78	Number of door switches (Safety supervisory function)
K79	Supervisory speed filtering time during servo-off (Safety supervisory function)
K80	Warning reference value (Vocal output)
K81	Timer for machining management
K82	Fixed value
K83	Maximum amount of compensation for workpiece thermal expansion
K84	Maximum allowable amount of compensation for workpiece thermal expansion in each interpolating cycle
K85	Special linear acceleration/deceleration time constant for threading
K87 - K89	Parameter for system internal setting Setting prohibited
K90	Return override during synchronous tapping
K91	Alternative M-code for M96
K92	Alternative M-code for M97
K93	Fixed value (2)
K94 0	Selection of whether menu function is to be made invalid after operation has been stopped by [ATC STOP]
K94 3	W-axis command code selection for the MDI mode
K95 0	Selection whether the difference in angle between the upper and lower turrets is to be incorporated into the current C-axis position display of HD2
K95 1	Fixed value (0)
K95 2	Tool position compensation during T-command execution, performed/not performed
K95 3	Coordinate system update during handle pulse interrupt, performed/not performed
K95 4	Fixed value (0)
K95 5	Acceleration/deceleration time constant for handle pulse feed
K95 6	Software limits for G30 execution valid/invalid
K95 7	In-position check valid/invalid
K96 0	G0 command in-position check valid/invalid
K96 1	Timing of manual free feed finish signal
K96 2	Fixed value (0)

Address (bit)	Outline
K96 3	Axis/Cutting interlock alarm display, valid/invalid
K96 4	Suppression of lost motion in modes other than the G1 command mode, valid/invalid
K96 5	Fixed value (0)
K96 6	Fixed value (0)
K96 7	Fixed G0 inclination
K97	B-axis misalignment correction, name of parallel axis
K98	B-axis misalignment correction, name of orthogonal axis
K99	Dynamic offset, name of rotational axis
K100	Dynamic offset, name of parallel axis
K101	Dynamic offset, name of orthogonal axis
K102 0	—
K102 1	—
K102 2	—
K102 3	Compensation method for the sensing tool's eccentricity and stylus radius in workpiece measurement and coordinate measurement, M Pro compatible/MATRIX scheme
K102 4	Z-axis direction for setting the TPC relay points of a measurement unit
K102 5	Selection of setup data (jaw data and tailstock position) to be used in EIA/ISO programs
K102 6	Setting the direction of rotation for milling tools mounted on the turret
K102 7	—
K103 0	Handling of the sign assigned to hobbing address D
K103 1	Synchronous control error auto-correction valid/invalid
K103 2	Fixed value
K103 3	DDB micron unit
K103 4	Fixed value
K103 5	Output of S-code and T-code at restart
K103 6	—
K103 7	Whether FIXED PT can be selected at RETURN of the end unit of the MAZATROL program
K104 0	Output of a laser measurement B-axis 0-degree command after ATC, valid/invalid

Address (bit)	Outline
K104 1	Direction of the laser axis of the laser measuring instrument (L16/K104 bit 2)
K104 2	Direction of the laser axis of the laser measuring instrument (Parallel to the Y-axis/X-axis)
K104 3	Voice Adviser (Vocal output function) valid/invalid
K104 4	Type of voice
K104 5	Vocal guidance for warm-up operation
K104 6	Rapid feed command when cutting feed override is set to 0%, operation is executed/alarm state
K104 7	Feed rate for an approach to the starting point in EIA modal restarting mode
K105 0	—
K105 1	Fixed value (1)
K105 2	S-code macro call invalid/valid
K105 3	T-code macro call invalid/valid
K105 4	Second auxiliary function macro call invalid/valid
K105 5	Fixed value (0)
K105 7	Fixed value (0)
K106 0	Execution conditions for user macroprogram interrupt
K106 1	Start timing for user macroprogram interrupt
K106 2	Fixed value (0)
K106 3	Fixed value
K106 4	Fixed value (0)
K106 5	Fixed value (0)
K106 6	Fixed value (0)
K106 7	Fixed value (0)
K107 0	Fixed value (0)
K107 5	Motion distance in the manual pulse feed mode for the axis whose position is displayed in diameter value
K107 6	Deceleration for arc valid/invalid
K107 7	Fixed value (0)
K108	Permissible error range for synchronous control

Address (bit)	Outline
K110	Judgment angle near a singular point
K111	Clamping speed in safety supervisory mode 3
K113	Machine type
K114	Axis number of the horizontal axis in the rectangular coordinate system
K115	Axis number of the vertical axis in the rectangular coordinate system
K116	Axis number of the height axis in the rectangular coordinate system
K117	Rotational direction of the rotary axis (Tool tip point control)
K121	Axis number of the first rotary axis
K122	Horizontal axis rotational center offset of the first rotary axis
K123	Vertical axis rotational center offset of the first rotary axis
K124	Height axis rotational center offset of the first rotary axis
K125	Axis number of the second rotary axis
K126	Horizontal axis rotational center offset of the second rotary axis
K127	Vertical axis rotational center offset of the second rotary axis
K128	Height axis rotational center offset of the second rotary axis

3. TABLE (L)

Address (bit)	Outline
L1	Stylus eccentricity of touch sensor (X-component)
L2	Stylus eccentricity of touch sensor (Y-component)
L3	Radius of stylus ball of touch sensor (X-component)
L4	Radius of stylus ball of touch sensor (Y-component)
L5	Z-axis stroke for tip position memory (TEACH function)
L6	Tool-breakage judgment distance for TBR function
L7	Tool-breakage restoration mode for TBR function (for M35 external tool breakage detection)
L8	Skipping stroke limit for MMS
L9	Selection of random ATC specifications
L10	Interval between magazine pockets

Address (bit)	Outline
L11	Touch sensor's interference direction
L12	Tolerance for manual measurement
L13	Allowable angle for parallelism and right angle in manual measurement
L14	Escapement for straightness measurement
L15	Macro program number for straightness measurement
L16	Parameter for system internal setting Setting prohibited
L17	Minimum angle of head indexing command (for INTEGREX j)
L18	Presence/absence of tailstock
L19	Number of electric steady rests
L21	Output type of index (rotary) table
L22	Data of the tool nose measurement sensor, Sensor width along the X-axis
L23	Data of the tool nose measurement sensor, Sensor width along the Z-axis
L24	Tool nose measurement sensor reference position, X-axis
L25	Tool nose measurement sensor reference position, Z-axis
L26	Tool nose measurement sensor reference position, Y-axis
L27	Timer setting for manual TOOL EYE measurement
L28	Amount of Z-axial retreat from the approach point for TOOL EYE measurement
L29	Machine efficiency
L30	Selection of machining navigation case introduction messages
L31 - L34	Use in machine manufacturer macro
L37	Minimum index angle of index table
L38	M/B-code for index of index table
L39	Selection of execution/non execution of indexing unit
L40	Availability of specification of index table angle in end unit
L41	Simultaneous operation of indexing unit with ATC
L42	Initial value of index table angle
L43	Indication of index table angle

Address (bit)	Outline
L44	Selection of automatic setting on/off for nose position correction of a drilling tool
L45	Index table angle command
L46	Maximum number of pallets in pallet changing unit
L47	To prepare or not to prepare next pallet change
L48	Number of long boring bars
L49	Simultaneous operation of pallet change with ATC
L50	Rewriting of head number
L51	Tool command system in MDI operation
L52	Writing of machining management data with macro variable
L53	Showing of program number in PALLET MANAGEMENT display
L54	Selection of automatic operation mode
L55	Spindle load meter display type
L56	Method of measurement of coordinates by tool edge memorizing function (TEACH)
L57	Rewriting of tool data during automatic operation
L58	Head index angle indication system
L59	Input selection for HEAD OFFSET display
L60	Head quantity
L61	Output timing of AHC and APC
L62	Head relay point X1
L63	Head relay point Y1
L64	Head relay point X2
L65	Head relay point Y2
L66	Return/No return to head indexing point Z
L67	Length between the end surface of the spindle and the center of head rotation
L68	Head correction value X
L69	Head correction value Y
L70	Axis movement from machining face on escapement

Address (bit)	Outline
L71	Shift of basic coordinate for oblique face machining
L73	Time constant for shape correction acceleration/deceleration filter 2
L74	Cutting feed rate for pre-interpolational acceleration/deceleration control
L75	Time constant for pre-interpolational linear control during cutting feed rate acceleration/deceleration
L76	Acceleration rate for high-speed cutting
L77	Angle for deceleration at corner before interpolation
L78	Amount of X-axial retreat from the approach point for TOOL EYE measurement
L79	In-position width for changeover of the synchronized-tapping gain
L80	Linear to arc deceleration rate
L81	Parameter for system internal setting Setting prohibited
L82	Table thickness
L83	Spindle head radius
L84	Correction value of alignment deviation X (Upper face)
L85	Correction value of alignment deviation Y (Upper face)
L86	Correction value of alignment deviation X (0-degree face)
L87	Correction value of alignment deviation Y (0-degree face)
L88	Correction value of alignment deviation X (90-degree face) Stylus eccentricity of touch sensor (X-component for B-axis at 90-degree position) (for INTEGREX series)
L89	Correction value of alignment deviation Y (90-degree face) Stylus eccentricity of touch sensor (Y-component for B-axis at 90-degree position) (for INTEGREX series)
L90	Correction value of alignment deviation X (180-degree face) Stylus eccentricity of touch sensor (X-component for B-axis at 180-degree position) (for INTEGREX series)
L91	Correction value of alignment deviation Y (180-degree face) Stylus eccentricity of touch sensor (Y-component for B-axis at 180-degree position) (for INTEGREX series)
L92	Correction value of alignment deviation X (270-degree face)
L93	Correction value of alignment deviation Y (270-degree face)
L94	X/Y travel distance during EIA-programmed tool measurement
L95	Offset number auto-setting for EIA-programmed tool measurement
L96	Amount of shifting during offset number auto-setting for EIA-programmed tool measurement
L97	M-code to be output when selecting menu function for the MDI tool designation
L98	Max. tool length for laser tool length measurement

Address (bit)	Outline
L99	Cycle time for saving the operational status management data
L100	Laser sensor position X
L101	Laser sensor position Y
L102	Laser sensor position Z
L103	Approach point X for laser tool diameter measurement
L104	Approach point Y for laser tool diameter measurement
L105	Approach point Z for laser tool diameter measurement
L106 0	Measuring equipment selection
L106 1	Selection of a rotational reference coordinate system for WPC-th
L106 2	Selection of measuring equipment
L106 3	Selection of whether workpiece measurement results and tool measurement results are to be stored into tool data of the lower turret
L106 4	Selection of whether to enable or disable the fixed amount compensation function
L106 5	B-axis user-defined angle setting in the coordinate measurement unit and workpiece measurement unit invalid/valid
L106 6	Non-compensation area for the tool measurement and execution timing of tool and workpiece measurement operation
L106 7	Selection of laser measuring menu display
L107 0	Tool path drawing
L107 1	Tailstock function valid/invalid
L107 2	Tailstock type
L107 3	Whether tail thrust is also to be displayed in pounds (lbs)
L107 4	LBB No. setting for the grooving I.D. tool, threading I.D. tool, or touch sensor, valid/invalid
L107 5	XYZ-axis operation for the first T-command after cycle start
L107 6	A-axis operation for the first T-command after cycle start
L107 7	Heavy tool designation valid/invalid
L108 0	Fixed value (0)
L108 1	Fixed value (0)
L108 2	Fixed value (0)
L108 3	Fixed value (0)

Address (bit)	Outline
L108 4	Fixed value (0)
L108 5	Incorporating the C-axis coordinate into the TOOL PATH CHECK , TRACE , and SHAPE CHECK displays of the VRX machine
L108 6	Fixed value (0)
L108 7	Displaying the milling tool nose in the MAZATROL program of the TOOL PATH CHECK display
L109 0	Axial direction of the two axes to be checked by the software limit relative position check function (1st set of axes)
L109 1	Axial direction of the two axes to be checked by the software limit relative position check function (2nd set of axes)
L109 2	Axial direction of the two axes to be checked by the software limit relative position check function (3rd set of axes)
L109 3	Axial direction of the two axes to be checked by the software limit relative position check function (4th set of axes)
L109 4	Formula selection for calculating horsepower and thrust on the TOOL DATA display with the [AUTO SET] menu function
L110 0	Opposed-spindle lathe specifications
L110 1	Vertically inverted spindle specifications
L110 2	Axis name of the secondary spindle
L110 3	Whether to disable or enable the display of "section to be machined" in the milling tool sequence
L110 4	RAM spindle specification
L110 5	Correction axis for LENGTH B
L110 6	—
L110 7	Z-axis direction
L111 5	T-code in the index unit to be executed before the pallet change unit
L112 0	Whether traction tailstock is present or absent
L112 1	Whether carriage-connection steadyrest 1 is present or absent
L112 2	Whether carriage-connection steadyrest 2 is present or absent
L112 3	Whether carriage-connection steadyrest 3 is present or absent
L112 4	Whether carriage-connection steadyrest 4 is present or absent
L112 5	—
L112 6	—
L112 7	—
L113	Software limit relative position check – Check reference axis (1st set of axes)

Address (bit)	Outline
L114	Software limit relative position check – Check reference axis (2nd set of axes)
L115	Software limit relative position check – Check reference axis (3rd set of axes)
L116	Software limit relative position check – Check reference axis (4th set of axes)
L117	Software limit relative position check – Check target axis (1st set of axes)
L118	Software limit relative position check – Check target axis (2nd set of axes)
L119	Software limit relative position check – Check target axis (3rd set of axes)
L120	Software limit relative position check – Check target axis (4th set of axes)
L121	Software limit relative position check – Limit distance in the approaching direction (1st set of axes)
L122	Software limit relative position check – Limit distance in the approaching direction (2nd set of axes)
L123	Software limit relative position check – Limit distance in the approaching direction (3rd set of axes)
L124	Software limit relative position check – Limit distance in the approaching direction (4th set of axes)
L125	Rotational axis prefiltering time constant
L126	Positioning direction of the head rotation - α -axis for oblique plane indexing
L129	Acceleration/deceleration filter (1st stage), G1 time constant
L130	Acceleration/deceleration filter (1st stage), G0 time constant
L131	Acceleration/deceleration filter (2nd stage), G1 time constant
L132	Acceleration/deceleration filter (2nd stage), G0 time constant
L133	<For tool tilt type> Tool rotational axis offset, horizontal axis offset amount for the 1st rotational axis
L134	<For tool tilt type> Tool rotational axis offset, vertical axis offset amount for the 1st rotational axis <For table tilt type> Y-axial runout of the A-axis
L135	<For tool tilt type> Tool rotational axis offset, height axis offset amount for the 1st rotational axis <For table tilt type> Z-axial runout of the A-axis
L136	<For tool tilt type> Tool rotational axis offset, horizontal axis offset amount for the 2nd rotational axis
L137	<For tool tilt type> Tool rotational axis offset, vertical axis offset amount for the 2nd rotational axis
L138	<For tool tilt type> Tool rotational axis offset, height axis offset amount for the 2nd rotational axis
L139	Minimum usable tool diameter of the measurable chamfering tool
L140	Waiting time for the completion signal of M35 (tool breakage detection) from PLC
L141	Software limit relative position check (for MR-J2-CT axis) – Task delay period
L142	Software limit relative position check (for MR-J2-CT axis) – MR-J2-CT gain

Address (bit)	Outline
L143	Software limit relative position check (for MR-J2-CT axis) – MR-J2-CT time constant
L144	Software limit relative position check (for MR-J2-CT axis) – MR-J2-CT rapid feed rate

4. FEED VEL. (M)

Address (bit)	Outline
M1	Rapid feed rate
M2	Feed rate for initial zero-point return
M3	Cutting feed rate limit
M4	Offset of machine coordinates system
M5	Second zero-point coordinating value
M6	Third zero-point coordinating value
M7	Fourth zero-point coordinating value
M8	Maximum software limit specified by manufacturer (+ direction)
M9	Maximum software limit specified by manufacturer (- direction)
M10	Command unit
M11	Coding of address of axis
M12	Coding of incremental axis
M13	Axis name (for display)
M14	Shifting distance of the watchdog-less home position
M15	Axis name (for axis name changing)
M16	Zero-point shift amount/grid shift amount
M17 0	—
M17 1	Unit of output from MCP to servo amplifier
M17 2	Direction of machine zero-point return
M17 3	Error correction schema with servo on
M17 4	Type of axis
M17 5	Rotational direction of servo motor (for movement in (+) direction)

Address (bit)	Outline
M17 6	—
M17 7	Alarm/No alarm with axis removed
M18 0	Spindle C-axis
M18 2	Machine zero-point position
M18 3	Virtual axis with dog
M18 4	Dog-less axis
M18 5	X-axis current position display in radius/diameter
M18 6	Automatic/manual simultaneous absolute-value updating
M18 7	Absolute-value detection
M19 0	Transmission of rapid feed rate value to SV69/SV70
M19 3	In-position checking method
M19 4	Parameter for system internal setting <u>Setting prohibited</u>
M19 5	Zero-point operation starting position check
M19 7	Backlash scheme to be adopted for dog-type returning to zero point
M20 0	Rotational direction of the rotation axis (Dynamic offset II)
M20 1	KOMET tool axis specification
M20 2	Method of jog/rapid feed stopping when the INTELLIGENT SAFETY SHIELD is valid
M20 6	Switching between zero-point shift amount and grid shift amount
M21 0	Rapid-feed acceleration/deceleration type, Linear acceleration/deceleration
M21 1	Rapid-feed acceleration/deceleration type, First-order lag
M21 2	Rapid-feed acceleration/deceleration type, Second-order lag
M21 3	Rapid-feed acceleration/deceleration type, Exponential acceleration/linear deceleration
M21 4	Cutting-feed acceleration/deceleration type, Linear acceleration/deceleration
M21 5	Cutting-feed acceleration/deceleration type, First-order lag
M21 6	Cutting-feed acceleration/deceleration type, Second-order lag
M21 7	Cutting-feed acceleration/deceleration type, Exponential acceleration/linear deceleration
M22 0	Deceleration time constant for rapid-feed exponential acceleration/linear deceleration

Address (bit)	Outline
M22 1	Type of stroke-end stop
M22 2	Type of stroke-end stop
M22 3	—
M22 4	—
M22 5	—
M22 6	—
M22 7	—
M23 0	Selection of NC control axes/PLC control axes
M25	Illegal axis area upper limits
M26	Illegal axis area lower limits
M27	Optimum acceleration control: Target speed
M29	Rapid feed clamping speed 1 for superposition control
M30	Rapid feed clamping speed 2 for superposition control
M31	Cutting feed clamping speed for superposition control
M32	Safety speed for Safety supervisory mode 2
M33	Safety speed for Safety supervisory mode 3
M34	Safety clamping speed for Safety supervisory mode 2
M35	Safety clamping speed for Safety supervisory mode 3
M36	Speed supervisory door selection
M37	Safety clamping speed reduction judgment coefficient
M38	Interference check distances for INTELLIGENT SAFETY SHIELD, primary check distance
M39	Interference check distances for INTELLIGENT SAFETY SHIELD, secondary check distance
M40	Ram spindle stored stroke end, upper limit
M41	Ram spindle stored stroke end, lower limit
M42	Interference check distance for INTELLIGENT SAFETY SHIELD during automatic operation
M45	Selection of a door for signal input on the drive side (servo) in safety supervisory mode

5. TIME CONST. (N)

Address (bit)	Outline
N1	Rapid-feed time constant (linear acceleration/deceleration)
N2	Cutting-feed time constant (linear acceleration/deceleration)
N3	Rapid-feed time constant (First-order lag)
N4	Time constant for post-interpolation rapid feed acceleration/deceleration filter
N5	Cutting-feed time constant (First-order lag)
N7	OT time
N8	Creeping speed during initial zero-point return
N9	Amount of grid ignorance during initial zero-point return
N10	Grid spacing
N12	Machine dependent specific value
N13	Machine dependent specific value
N15	Machine dependent specific value
N16	Machine dependent specific value
N17	Servo amplifier channel number
N18	Servo amplifier rotary switch number
N19	Axis system number
N21 0	Linear-type rotational axis
N21 1	Rotational axis shortcut Invalid/Valid
N21 2	Fixed value (0)
N21 3	Fixed value
N21 4	Reference axis for superposition control
N21 5	Superposition axis for superposition control
N21 6	Relative polarity of control axis
N21 7	Current position immediately after cross machining
N22 0	Axis selection for 3D interference check during automatic operation and for positioning on the VIRTUAL MACHINING display
N25	Time constant for deceleration rate calculation
N26	Accuracy coefficient for deceleration rate calculation

Address (bit)	Outline
N27	Rapid feed time constant for superposition
N28	Cutting feed time constant for superposition
N29	Time constant for shape correction rapid feed acceleration/deceleration filter
N30	Time constant for cutting feed (for M881)
N31	Time constant for cutting feed (for M882)
N32	Time constant for cutting feed (for M883)
N33	Time constant for cutting feed (for M884)
N34	Time constant for cutting feed (for M885)
N35	Time constant for cutting feed (for M886)

6. ANOTHER (S)

Address (bit)	Outline
S3	Feed forward gain for the MAZAK Precision Rapid Boring Tornado Option
S4	Feed forward gain
S5	Rotational center of the table
S6	Absolute position detection parameter
S7	Upper limit (on Z-axis) of machining range for table rotating machining I
S8	Feed-forward gain G00
S10	Axis of rotation of the tilting table
S11	Corner position of the tilting table
S12	Axis of rotation of the tilting table (Used for the automatic program origin calculation function)
S13	G00 in-position width
S14	G01 in-position width
S15	Fixed value
S16	Unbalanced axis torque offset
S17	Torque limit buffer reduction ratio 1
S18	Torque limit buffer reduction ratio 2
S19	Tool change completion position of the long boring bar end tool

Address (bit)	Outline
S20	APC completion position
S21	Completion position of gantry robot operation
S22	Cutting feed clamping speed during tool tip point control
S23	Reference workpiece zero point
S25	Central position on the upper surface of the tilt table (VARIAXIS) Central position of table upper surface (other machining centers)
S26	Central position of tilt table rotation (VARIAXIS)
S27	Ram spindle offset amount
S28	Center of the workpiece to be directly mounted on the table of the machines equipped with an additional table
S29	Position of the work hand rotating axis in relation to the machine zero point
S30	Distance from the axis of rotation of the work hand to the reference position of the vice for the work hand
S31	Specification of the machine coordinates of the floating reference point
S32	Rapid traverse speed for the mode of tool tip point control
S33	Reference position X for the TEACH function, to be used for setting the LENGTH B item of an angular tool (For machines of five-surface machining by angular tools)
S35	Rotational center of the second table
S38	Time-out period for releasing the emergency stop status during powering on

7. SPINDLE (SA)

Address (bit)	Outline
SA1 - SA8	Maximum spindle speed in each speed range (range 1 to 8)
SA9 - SA16	Constants for calculating each gear speed of the spindle (range 1 to 8)
SA17 - SA24	Maximum spindle speed during tapping cycle (range 1 to 8)
SA25 - SA28	Spindle speed during gear shifting (range 1 to 4)
SA29	Load meter reference output scaling factor (high-speed coil)
SA30	Load meter reference output base rotational speed (high-speed coil)
SA31	Load meter reference output scaling factor (low-speed coil)
SA32	Load meter reference output base rotational speed (low-speed coil)
SA33 - SA40	Acceleration/deceleration time constant for the synchronous tapping (range 1 to 8)
SA41	Spindle orientating speed

Address (bit)	Outline
SA42	Minimum spindle speed
SA43	Channel number for the spindle amplifier
SA44	Spindle amplifier rotary switch number
SA45 0	Spindle speed range changing method, in relation to switching the torque factors for auto-pecking of the cutting load detection type
SA45 1	Spindle speed range changing method, in relation to switching the torque factors for auto-pecking of the cutting load detection type 2
SA45 2	Homing direction for synchronous tapping
SA45 3	Homing direction for synchronous tapping
SA45 4	Defines the specified direction as the Z-phase detection direction
SA45 5	Spindle index gear correction
SA46 0	Direction of orientation
SA46 1	Direction of orientation
SA46 2	C-axis position control changeover type
SA46 3	Synchronous tapping position control changeover type
SA46 4	Z-phase detection direction
SA46 5	C-axis homing direction
SA46 6	C-axis homing direction
SA46 7	Synchronous tapping command polarity
SA47 0	—
SA47 1	—
SA47 2	—
SA47 3	—
SA47 4	—
SA47 5	—
SA47 6	—
SA47 7	Considering/Ignoring the spindle/motor gear ratio
SA48	Encoder signal input destination
SA49	Speed attainment detection width

Address (bit)	Outline
SA50	Spindle type
SA51	Number of gears on spindle
SA52	Turning spindle type
SA53	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — L coils
SA54	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — H coils
SA55	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — Spindle gear position 3
SA56	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — Spindle gear position 4
SA57	Spindle viscous friction coefficient "cms" for auto-pecking of the cutting load detection type — Range 1
SA58	Spindle viscous friction coefficient "cms" for auto-pecking of the cutting load detection type — Range 2
SA59	Spindle viscous friction coefficient "cms" for auto-pecking of the cutting load detection type — Range 3
SA60	Spindle viscous friction coefficient "cms" for auto-pecking of the cutting load detection type — Range 4
SA61	Spindle coulombic friction coefficient "fms" for auto-pecking of the cutting load detection type — Range 1
SA62	Spindle coulombic friction coefficient "fms" for auto-pecking of the cutting load detection type — Range 2
SA63	Spindle coulombic friction coefficient "fms" for auto-pecking of the cutting load detection type — Range 3
SA64	Spindle coulombic friction coefficient "fms" for auto-pecking of the cutting load detection type — Range 4
SA65	Cutting force calculation filter for auto-pecking of the cutting load detection type
SA66	Maximum permissible speed of the rotational axis for polygonal/hobbing machining
SA67 - SA73	Rotational speed in the following spindle output diagrams: MACHINING NAVIGATION-RESULT/MACHINING NAVIGATION-PPEDITION/Monitoring Functions/POSITION
SA74 - SA80	Output in the following spindle output diagrams MACHINING NAVIGATION-RESULT/MACHINING NAVIGATION-PPEDITION/Monitoring Functions/POSITION
SA81	Spindle limit speed selection for spindle position control time constants — Limit speed 1
SA82	Spindle limit speed selection for spindle position control time constants — Limit speed 2
SA83	Spindle limit speed selection for spindle position control time constants — Limit speed 3
SA84	Spindle position control time constants — Time constant 1
SA85	Spindle position control time constants — Time constant 2
SA86	Spindle position control time constants — Time constant 3
SA87	Spindle speed operating time constant changeover revolutions 1
SA88	Spindle speed operating time constant changeover revolutions 2

Address (bit)	Outline
SA89	Spindle speed operating time constant 1
SA90	Spindle speed operating time constant 2
SA91	Spindle speed operating time constant 3
SA92	Z-phase detection speed
SA93	Amount of synchronous tapping zero point shifting
SA94	Homing speed for synchronous tapping
SA95	Maximum revolutions in manual operation mode
SA96	Amount of orientation position shifting
SA97	Reduction ratio of the synchronous tapping time constant for high-speed synchronous tapping
SA99	Orientation time constant
SA100 - SA106	Rotational speed in the following spindle output diagrams at continuous rating: MACHINING NAVIGATION-RESULT/MACHINING NAVIGATION-PPEDITION/Monitoring Functions/POSITION
SA107 - SA113	Output in the following spindle output diagrams at continuous rating: MACHINING NAVIGATION-RESULT/MACHINING NAVIGATION-PPEDITION/Monitoring Functions/POSITION
SA114	Spindle speed supervisory mode 2 for safety supervision
SA115	Spindle speed supervisory mode 3 for safety supervision
SA116	Spindle safety clamping mode 2 for safety supervision
SA117	Spindle safety clamping mode 3 for safety supervision
SA118	Selecting the spindle door of the spindle whose speed is to be monitored
SA119	Deceleration judgment coefficient on safety speed clamping of the spindle to be supervised for safety
SA120	PLG pulse rate for spindle index gear tooth correction
SA121	Amount of branching [1] point correction for spindle index gear tooth
SA122	Amount of branching [2] point correction for spindle index gear tooth
SA123	Amount of branching [3] point correction for spindle index gear tooth
SA124	Amount of branching [4] point correction for spindle index gear tooth
SA125	Amount of branching [5] point correction for spindle index gear tooth
SA126	Amount of branching [6] point correction for spindle index gear tooth
SA127	Amount of branching [7] point correction for spindle index gear tooth
SA128	Amount of branching [8] point correction for spindle index gear tooth

Address (bit)	Outline
SA129	#1 notch filter frequency for cutting-load detection pecking cycle
SA130	#1 notch filter depth for cutting-load detection pecking cycle
SA131	#2 notch filter frequency for cutting-load detection pecking cycle
SA132	#2 notch filter depth for cutting-load detection pecking cycle
SA133	Breakage detection torque for cutting-load detection pecking cycle
SA134	Maximum spindle orientation speed
SA136	Selection of signal input door on the safety monitor drive side (spindle)
SA138	Number of spindle gears
SA139	Number of motor gears
SA140	Turret indexing gear ratio
SA143 1	Waiting for phase compensation completion at the start of high-speed synchronous tapping
SA143 3	Spindle speed for the orient command given during spindle rotation
SA144 4	Positive threading spindle rotational direction and encoder polarity
SA144 5	Gear conversion during spindle rotation valid
SA144 6	Spindle gear changeover valid
SA144 7	Turret indexing valid

8. BARRIER (BA)

Address (bit)	Outline
BA1	Chuck outside diameter (for chuck barrier) — No. 1 turning spindle
BA2	Chuck width (for chuck barrier) — No. 1 turning spindle
BA3	Chuck inside diameter (for chuck barrier) — No. 1 turning spindle
BA4	EIA program workpiece outside diameter
BA5	Chuck outside diameter (for chuck barrier) — No. 2 turning spindle
BA6	Chuck width (for chuck barrier) — No. 2 turning spindle
BA7	Chuck inside diameter (for chuck barrier) — No. 2 turning spindle
BA8	Tail body outside diameter (for tail barrier)
BA9	Tail body length (for tail barrier)

Address (bit)	Outline
BA10	Tail spindle outside diameter (for tail barrier)
BA11	Length with tail spindle at back end (for tail barrier)
BA12	Tail head outside diameter (for tail barrier)
BA13	Tail head length (for tail barrier)
BA14	Tail head taper angle (for tail barrier)
BA15	Tail head biting diameter (for tail barrier)
BA16	Tail barrier, tail extruding length
BA17	Tail barrier, workpiece length
BA18	Tail barrier, tail reversing position Z
BA19	Distance from the Z-axis machine zero point to the spindle edge — No. 1 turning spindle
BA20	Distance from the Z-axis machine zero point to the spindle edge — No. 2 turning spindle
BA21	Jaw number for EIA program barrier — No. 1 turning spindle
BA22	Jaw number for EIA program barrier — No. 2 turning spindle
BA23	Turret outside diameter (turret type tool post)
BA24	Turret width (turret type tool post)
BA25	Turret reference position X (turret type tool post)
BA26	Turret reference position Z (turret type tool post)
BA27	Tool holder mounting position — Type 1
BA28	Tool holder width in X-axis direction — Type 1
BA29	Tool holder width in Z-axis direction — Type 1
BA30	Tool holder mounting position — Type 2
BA31	Tool holder width in X-axis direction — Type 2
BA32	Tool holder width in Z-axis direction — Type 2
BA33	Tool holder mounting position — Type 3
BA34	Tool holder width in X-axis direction — Type 3
BA35	Tool holder width in Z-axis direction — Type 3
BA36	Tool holder mounting position — Type 4

Address (bit)	Outline
BA37	Tool holder width in X-axis direction — Type 4
BA38	Tool holder width in Z-axis direction — Type 4
BA39	EIA tool barrier, tool holder mounting position
BA40	EIA tool barrier, tool holder width X
BA41	EIA tool barrier, tool holder width Z
BA42	Barrier type
BA43	First tool number (in the 1st set of tools)
BA44	Number of tools (in the 1st set of tools)
BA45	First tool number (in the 2nd set of tools)
BA46	Number of tools (in the 2nd set of tools)
BA47	Turret type
BA48	Axis name of the head to be rotated
BA49	Axis number of the inclined axis
BA50	Fundamental axis number
BA51	Inclined-axis control, vector of virtual Y
BA52	Inclined-axis control, vector of real X
BA53	Inclined-axis control, vector of real Y
BA54	Selection of work spindle for hobbing
BA55	Turning spindle number for polygonal machining (D1)
BA56	Turning spindle number for polygonal machining (D2)
BA57	Turning spindle number for polygonal machining (D3)
BA58	Turning spindle number for polygonal machining (D4)
BA59	Spindle forward rotation M-code for tapping cycle
BA60	Spindle reverse rotation M-code for tapping cycle
BA61	Amount of runout of the B-axis center
BA62	Amount of offset for the B-axis — spindle distance
BA63	Holder angle of angle tool holder

Address (bit)	Outline
BA64	B-axis tool reference position X
BA65	B-axis tool reference position Z
BA66	Deceleration area Z
BA67	Measuring area Z
BA68	Deceleration area X
BA69	Measuring area X
BA70	Distance between the reference points on both turrets
BA71	System number to be used when argument L is omitted from G112
BA72	Feed override for retracting the tap
BA73	Barrier valid/invalid 1 (chuck, sub-chuck, tailstock)
BA74	Barrier valid/invalid 2 (Lower turret, steady rest)
BA75 - BA78	Barrier setup turret reference position
BA79 - BA82	Barrier setup chuck reference position 1
BA83 - BA86	Barrier setup chuck reference position 2
BA87 - BA90	Barrier reference position for a powered tailstock
BA91	Distance from spindle edge to partition plate
BA92	Central position X when viewed from machine zero point
BA93	Upper/lower turret tool angle difference (HD1)
BA94	Upper/lower turret tool angle difference (HD2)
BA95	Tool nose measurement, sensor width along the X-axis
BA96	Tool nose measurement, sensor width along the Z-axis
BA97	Tool nose measurement, X-coordinate of the sensor's reference point
BA98	Tool nose measurement, Y-coordinate of the sensor's reference point
BA99	Tool nose measurement, Z-coordinate of the sensor's reference point
BA100	Tool nose measurement, X-coordinate of the sensor's reference point (for lower turret/HD2)
BA101	Tool nose measurement, Y-coordinate of the sensor's reference point (for lower turret/HD2)
BA102	Tool nose measurement, Z-coordinate of the sensor's reference point (for lower turret/HD2)

Address (bit)	Outline
BA103	Tool nose measurement, sensor width along the X-axis (for lower turret/HD2)
BA104	Tool nose measurement, sensor width along the Z-axis (for lower turret/HD2)
BA105	Adjustment "FdT" for servo axis thrust hold
BA106	Dead zone
BA107	Filter
BA108	Servo axis thrust hold droop
BA109	Offset amount (X) during automatic tool setting value calculation
BA110	Offset amount (Z) during automatic tool setting value calculation
BA111	Pre-interpolation acceleration/deceleration time constant for time constant changeover M-code (M881)
BA112	Pre-interpolation acceleration/deceleration time constant for time constant changeover M-code (M882)
BA113	Pre-interpolation acceleration/deceleration time constant for time constant changeover M-code (M883)
BA114	Pre-interpolation acceleration/deceleration time constant for time constant changeover M-code (M884)
BA115	Pre-interpolation acceleration/deceleration time constant for time constant changeover M-code (M885)
BA116	Pre-interpolation acceleration/deceleration time constant for time constant changeover M-code (M886)
BA117	Outside diameter of the turret for Virtual Machining
BA118	Width of the turret for Virtual Machining
BA119	Selecting a head having a lower-turret TOOL EYE
BA120	Mounting position X for an assembly tool model
BA121	Mounting position Z for an assembly tool model
BA122	Turret reference position X (for VIRTUAL MACHINING or 3D MONITOR display)
BA123	Turret reference position Z (for VIRTUAL MACHINING or 3D MONITOR display)
BA124	Fixed value (0)
BA125 0	EIA tailstock barrier, tailstock usage valid/invalid
BA125 1	Tailstock present/absent
BA125 2	EIA tool barrier, tool holder present/absent
BA125 3	Barrier tool nose position (EIA, automatic)
BA125 4	Setting of BA11 is handled as tailstock extruding length, valid/invalid

Address (bit)	Outline
BA125 5	Output of spindle revolution M-code specified in same block as synchronous tapping, valid/invalid
BA125 6	Alarm stop when [BARRIER CANCEL] menu function is valid at the start of the automatic operation, valid/invalid
BA125 7	Alarm stop when jaw Nos. are not specified at the start of the automatic operation, valid/invalid
BA126 0	System to be made valid without system selection for queuing
BA126 1	Y-axis interference type
BA126 2	Y-axis moving range display
BA126 3	Tool command scheme
BA126 4	FLASH tool valid/invalid
BA126 5	Milling spindle orientation command (M219), output/no output
BA126 6	System without turret
BA126 7	Shape of the tailstock center for Virtual Machining, single stage/twin stage
BA127 0	M249 command for preparing for tool change before moving the tool to the tool change position using a MAZATROL program, output/no output
BA127 1	For programs of Z-offset scheme: Fixed value (1) For other programs: Invalid
BA127 2	Simultaneous output of TTM6M200 and TTM6M202
BA127 3	NC tailstock (servo axis thrust control)
BA127 4	M90/M91 output for machines with turret
BA127 5	Type of the clamping and unclamping M-codes that the NC automatically output for machines equipped with a spindle head of swiveling type
BA127 6	—
BA127 7	Whether or not to read the current settings (tailstock position) on the SET UP MANAG. display at the beginning of the execution of each program
BA128 0	NC tailstock pushing direction
BA128 1	Method of automatic determination of the virtual nose point for a tool measuring unit
BA128 2	Display of THRUST F. and HORSE PW for the AFC function in the first set of tool data (BA43, BA44)
BA128 3	Display of THRUST F. and HORSE PW for the AFC function in the second set of tool data (BA45, BA46)
BA128 6	Turret lathe with an ATC unit
BA128 7	Display of [TURRET UNCLAMP] for machines with an ATC-type turret
BA129 1	Fixed value
BA129 2	Fixed G0 inclination control in the G68/G68.5 mode

Address (bit)	Outline
BA129 3	Selection of whether or not to measure the milling tool length with the tool measuring unit
BA129 4	Display of [BARRIER CANCEL] menu item with [BARRIER CANCEL TEACH] made valid
BA129 5	Handling of measurement speed of 1 in/min or higher during laser tool length/diameter measurement with inch-specification machines
BA131 0	Multi-magazine number display

2-2-3 Data I/O parameter

1. CMT parameter (CMT)

Address (bit)	Outline
CMT1 - CMT24	—
CMT25	Type of processing to be executed if the tool quantity data within the NC memory mismatches that of the CMT
CMT26 - CMT32	—

2. TAPE parameter (TAP)

Address (bit)	Outline
TAP1	Type of terminator
TAP2	Terminator code 1
TAP3	Terminator code 2
TAP4	Output of CR during ISO code punching
TAP5	DC code parity
TAP6	Feed section DC code output
TAP7	DSR output flow control selection
TAP9	"[" code for paper tape reader/puncher for EIA
TAP10	
TAP11	"#" code for paper tape reader/puncher for EIA
TAP12	"*" code for paper tape reader/puncher for EIA
TAP13	"=" code for paper tape reader/puncher for EIA
TAP14	" :" code for paper tape reader/puncher for EIA
TAP15	"(" code for paper tape reader/puncher for EIA
TAP16	")" code for paper tape reader/puncher for EIA
TAP24	Tape operation port selection
TAP25	Paper tape puncher parity-V check
TAP26	Bit parameter related to paper tape reader/puncher
TAP27	Bit parameter related to program end code (M) for paper tape reader

Address (bit)	Outline
TAP29	Number of characters in feed section for paper tape puncher
TAP30	Number of characters in the space between O-number and program for paper tape puncher
TAP31	Number of characters in the space between programs for paper tape puncher

3. DNC parameter (DNC)

Address (bit)	Outline
DNC1	Type of terminator
DNC2	Terminator code 1
DNC3	Terminator code 2
DNC5	DC code parity
DNC9	Number of NC transmission retries during DNC file transfer
DNC10	Number of NC reception retries during DNC file transfer
DNC11	Number of NC transmission/reception retries during DNC command message transfer
DNC12	@ waiting time during DNC transmission
DNC13	"*", TEXT waiting time during DNC transmission
DNC14	EOT waiting time during DNC transmission
DNC15	NC stop time after reception of !
DNC16	NC reset time after digital-out
DNC17	NC stop time from reception
DNC18	DNC command reply message waiting time
DNC19	DNC machine number
DNC20	NC transmission stop time of DNC (from reception to transmission)
DNC21	NC transmission stop time of DNC (from transmission to transmission)
DNC25 0	Type of processing to be executed if the tool quantity data within the NC memory mismatches that which has been transferred from the DNC memory
DNC25 1	—
DNC25 2	Handling of tool data and tool files in the M PLUS format, valid/invalid
DNC25 3	—
DNC25 4	—

Address (bit)	Outline
DNC25 5	—
DNC25 6	—
DNC25 7	—
DNC26 0	After program reception, a search is made/not made for the work number of that program.
DNC26 1	Details of an alarm occurring in DNC are displayed or not.
DNC26 2	Loading of programs having the same work number as that of the registered program in NC becomes impossible or not.
DNC26 3	The function of the PROGRAM LOCK/ ENABLE switch is released or not.
DNC26 4	—
DNC26 5	Three digit G-format and G10 format codes input/output for MAZAK data transfer protocol
DNC26 6	Binary to ASCII format input/output of MAZAK data transfer protocol
DNC26 7	All programs having work numbers smaller than No. 9000 are erased/not erased at the start of program reception.
DNC29	Number of retry times with detection of a physical error
DNC30	Tool data/tool file message format
DNC31	Waiting time for a response from the COM port driver

4. Other (IOP/DPR/IDD)

Address (bit)	Outline
IOP5 1	Tool data count check during tool data loading
IOP5 6	Overwriting associated with loading of the program of the same work number
IOP5 7	Overwriting associated with loading of the tool model of the same name
IOP6 0	—
IOP6 1	—
IOP6 2	—
IOP6 3	—
IOP6 4	—
IOP6 5	—
IOP6 6	Selection between M640T and M640MT for loading MAZATROL programs in text file format
IOP6 7	—

Address (bit)	Outline
IOP7	Data entry for communication with the magazine-side display unit
IOP8 0	(Magazine-side display unit IN/OUT) Tool count/time data clear setting
IOP8 1	(Magazine-side display unit IN/OUT) Tool life count/time data clear setting
IOP9	Fixed value (0)
IOP14	MAZA-CARE function
DPR1	Baud rate
DPR2	Stop bit
DPR4	Data bit
DPR8	ISO code CR output and the output file size
DPR9	Method of handshaking
DPR10	DC code parity
DPR11	Feed section DC code output
DPR12	Waiting time
DPR13	Output format
DPR14	Selection of an output destination port
DPR15	Number of characters or the number of lines in feed section
DPR16	DPRNT operation port selection
IDD1 - IDD16	—

2-3 Detailed Description

2-3-1 Structure of the parameter list

Each parameter list is written in the following format:

Classification	[1]	Display title	[2]
Address	Meaning		Description
[3]	[4]		[9]
	Program type	[5]	
	Conditions	[6]	
	Unit	[7]	
	Setting range	[8]	

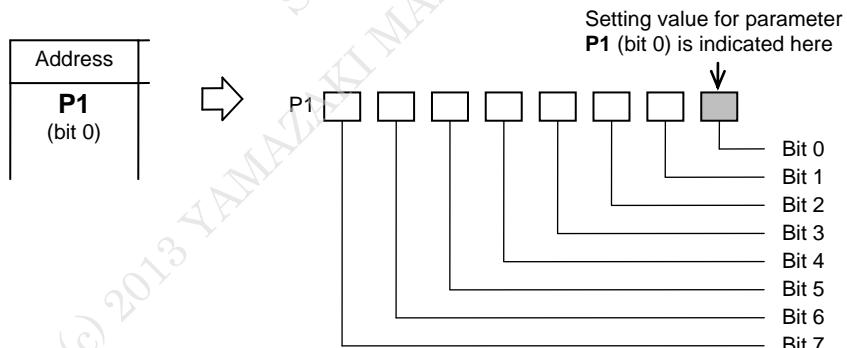
[1] Classification of parameters (USER, MACHINE or DATA I/O)

[2] Characters displayed at the upper part of the screen

[3] Parameter address displayed on the screen

- Bit input type parameters have the bit No. shown in the parentheses below address.

Example :



[4] Meaning of the parameter

[5] Applicable program

M Effective only for MAZATROL programs

E Effective for EIA/ISO programs

M, E Effective for MAZATROL programs and EIA/ISO programs

- [6] Conditions under which a changed parameter becomes valid

Example 1: "Immediate" designates that new parameter value becomes effective upon parameter change.

Example 2: "At power on" designates that new parameter value will become effective after procedure below.

1. Change parameter setting value.
↓ (By procedure similar to changing of ordinary data)
2. Press power off button on the operation panel.
↓
3. Press power on button on the operation panel.

Example 3: If "At I/O startup" is specified

The system operates at the parameter data entered before the start of I/O. If the parameter data is modified during I/O operation, the new data will not become valid until the I/O operation has been completed.

Example 4: If "After stop of movement" is specified

When the movement of the machine is completed, the program will operate with the new parameter value. Changing the parameter during the machine movement will not make the new parameter value valid during the movement.

Example 5: If "Next block" is specified

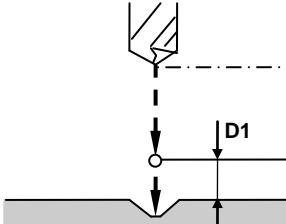
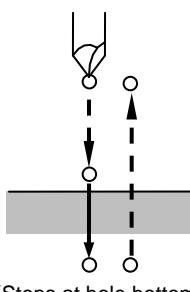
When the next block is loaded, the program will operate with the new parameter value. Changing the parameter while the block is being loaded will not make the new parameter value valid in that block.

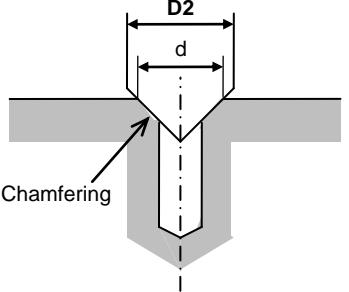
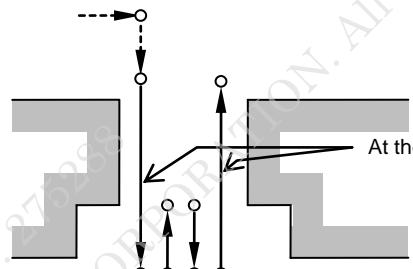
- [7] Units of data displayed

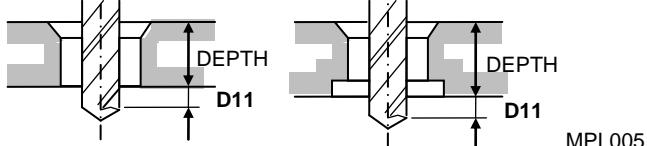
- [8] Allowable range of data (for an effective data range, refer to "Description" column.)

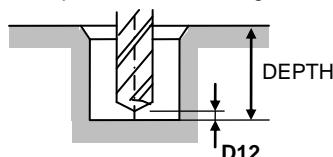
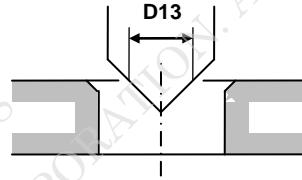
- [9] Details or meaning of the parameter

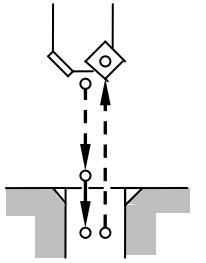
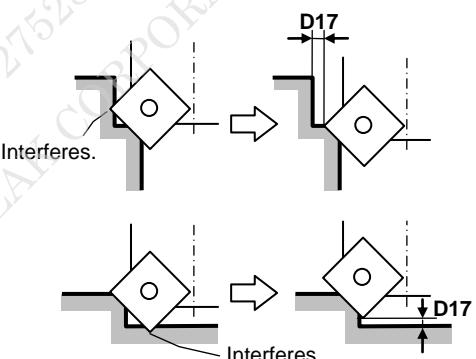
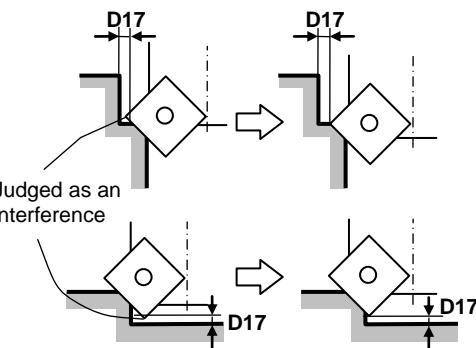
2-3-2 User parameter POINT (D)

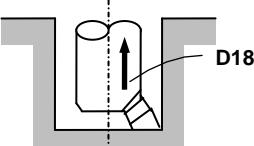
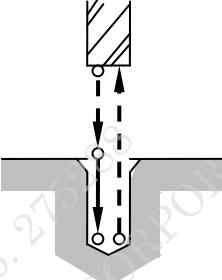
Classification	USER	Display title	POINT																								
Address	Meaning	Description																									
D1	Height of the second R-point during point machining	<p>Height of the second R-point</p>  <p>MPL001</p> <p>The height of the R-point during point machining is basically D41, however, it is changed to D1 under the following conditions.</p> <table border="1"> <thead> <tr> <th>Tool sequence</th> <th>Conditions</th> </tr> </thead> <tbody> <tr> <td>Drill</td> <td> <ul style="list-style-type: none"> Bit 6 of parameter D91 is set to 1 (D1 valid). There is a spot drill in the pre-machining tool sequence of the same unit. </td> </tr> <tr> <td>Reamer</td> <td> <ul style="list-style-type: none"> Bit 2 of parameter D92 is set to 1 (D1 valid). There is a chamfering cutter in the pre-machining tool sequence of the same unit. </td> </tr> </tbody> </table> <p>However, when a drill is included in the pre-machining tool sequence in case of a drilling tool sequence, the height is changed to D42. (⇒D42)</p>		Tool sequence	Conditions	Drill	<ul style="list-style-type: none"> Bit 6 of parameter D91 is set to 1 (D1 valid). There is a spot drill in the pre-machining tool sequence of the same unit. 	Reamer	<ul style="list-style-type: none"> Bit 2 of parameter D92 is set to 1 (D1 valid). There is a chamfering cutter in the pre-machining tool sequence of the same unit. 																		
Tool sequence	Conditions																										
Drill	<ul style="list-style-type: none"> Bit 6 of parameter D91 is set to 1 (D1 valid). There is a spot drill in the pre-machining tool sequence of the same unit. 																										
Reamer	<ul style="list-style-type: none"> Bit 2 of parameter D92 is set to 1 (D1 valid). There is a chamfering cutter in the pre-machining tool sequence of the same unit. 																										
D2	Nominal diameter of spot-machining tool	<p>The nominal diameter of a spot-machining tool that is automatically set during automatic tool development.</p> <p>Example :</p> <table border="0"> <tr> <td>SNo.</td> <td>TOOL</td> <td>NOM-Φ</td> <td>No.</td> <td>HOLE-Φ</td> <td>HOLE-DEP</td> </tr> <tr> <td>1</td> <td>CTR-DR</td> <td>(20.)</td> <td></td> <td>10.</td> <td>◆</td> </tr> <tr> <td></td> <td></td> <td>↑</td> <td></td> <td></td> <td></td> </tr> <tr> <td></td> <td></td> <td>D2</td> <td></td> <td></td> <td></td> </tr> </table>		SNo.	TOOL	NOM-Φ	No.	HOLE-Φ	HOLE-DEP	1	CTR-DR	(20.)		10.	◆			↑						D2			
SNo.	TOOL	NOM-Φ	No.	HOLE-Φ	HOLE-DEP																						
1	CTR-DR	(20.)		10.	◆																						
		↑																									
		D2																									
D3	Number of revolutions during dwell at hole bottom in spot-machining cycle	Z-axis feed dwell time at the hole bottom in a spot-machining cycle. Set this time in spindle revolutions.	<p>When the spot-machining tool reaches the hole bottom, the Z-axis will firstly stop moving until the spindle makes D3 revolutions, and then return to the original position at the rapid feed rate.</p>  <p>MPL002</p>																								

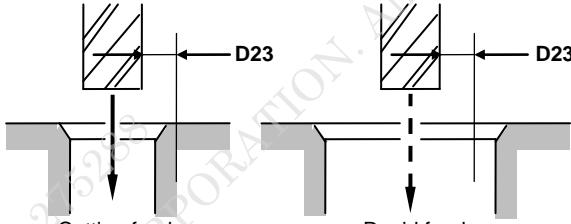
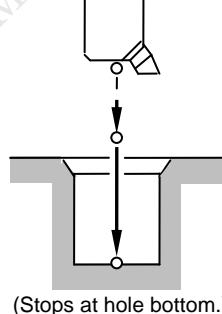
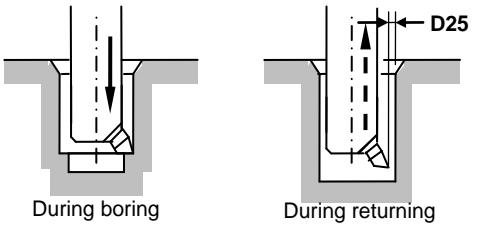
Classification	USER	Display title	POINT						
Address		Meaning							
D4 Maximum allowable spot-chamfering hole diameter element		Element used to set the maximum spot-chamfering hole diameter (d) during automatic tool development							
									
		Program type M							
		Conditions Immediate							
		Unit 0.1 mm/0.01 in							
D5 Prehole through speed during inversed spot-facing		Setting range 0 to 99 The feed rate of a tool as it is being passed through the prehole during an inversed spot-facing cycle Note : 0.5 mm/rev if this parameter setting is 0.							
									
		Program type M							
		Conditions Immediate							
		Unit 100 mm/min / 10 in/min							
D6 D7 Drill-machining cycle setting element		Setting range 0 to 9 Element used to automatically set drill-machining cycles during automatic tool development							
		<table border="1"> <thead> <tr> <th>Machining cycle</th> <th>Conditions</th> </tr> </thead> <tbody> <tr> <td>Drilling cycle</td> <td>$\frac{\text{DEPTH}}{\text{DIA}} \leq \text{D6}$</td> </tr> <tr> <td>High-speed deep-hole drilling cycle</td> <td>$\text{D6} < \frac{\text{DEPTH}}{\text{DIA}} \leq \text{D7}$</td> </tr> <tr> <td>Deep-hole drilling cycle</td> <td>$\text{D7} < \frac{\text{DEPTH}}{\text{DIA}}$</td> </tr> </tbody> </table>		Machining cycle	Conditions	Drilling cycle	$\frac{\text{DEPTH}}{\text{DIA}} \leq \text{D6}$	High-speed deep-hole drilling cycle	$\text{D6} < \frac{\text{DEPTH}}{\text{DIA}} \leq \text{D7}$
Machining cycle	Conditions								
Drilling cycle	$\frac{\text{DEPTH}}{\text{DIA}} \leq \text{D6}$								
High-speed deep-hole drilling cycle	$\text{D6} < \frac{\text{DEPTH}}{\text{DIA}} \leq \text{D7}$								
Deep-hole drilling cycle	$\text{D7} < \frac{\text{DEPTH}}{\text{DIA}}$								
Program type M									
Conditions Immediate									
Unit —									

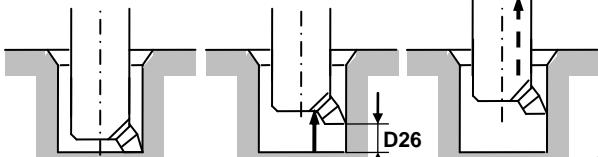
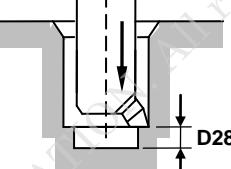
Classification	USER	Display title	POINT									
Address	Meaning	Description										
D8	Maximum diameter of holes machinable on one drill		Element used to automatically set the number of drills which are automatically developed according to the hole diameter of the drill unit									
			<table border="1"> <thead> <tr> <th>Number of drills developed</th> <th>Conditions</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>DIA ≤ D8</td> </tr> <tr> <td>2</td> <td>D8 < DIA ≤ D9</td> </tr> <tr> <td>3</td> <td>D9 < DIA ≤ D10</td> </tr> <tr> <td>Alarm</td> <td>D10 < DIA</td> </tr> </tbody> </table>	Number of drills developed	Conditions	1	DIA ≤ D8	2	D8 < DIA ≤ D9	3	D9 < DIA ≤ D10	Alarm
Number of drills developed	Conditions											
1	DIA ≤ D8											
2	D8 < DIA ≤ D9											
3	D9 < DIA ≤ D10											
Alarm	D10 < DIA											
Program type	M											
Conditions	Immediate											
Unit	mm/0.1 in											
Setting range	0 to 99											
D9	Maximum diameter of holes machinable on two drills											
	Program type	M										
	Conditions	Immediate										
	Unit	mm/0.1 in										
	Setting range	0 to 99										
D10	Maximum diameter of holes machinable on three drills											
	Program type	M										
	Conditions	Immediate										
	Unit	mm/0.1 in										
	Setting range	0 to 99										
D11	Through-hole/tap-prehole machining overshoot		Element used to automatically set the hole-drilling, endmilling, and boring depths during automatic tool development of inverted spot-facing, tapping, back-boring, through-hole drilling, through-hole counter-boring, and spot-faced tapping units									
			 MPL005									
			Example :									
			SNO. TOOL NOM- ϕ No. HOLE- ϕ HOLE-DEP 1 CTR-DR 10. 10. (21.) ↑ (DEPTH + D11)									
	Program type	M										
	Conditions	Immediate										
	Unit	0.1 mm/0.01 in										
	Setting range	0 to 99										

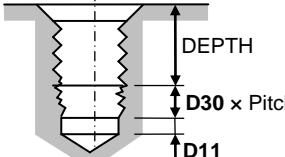
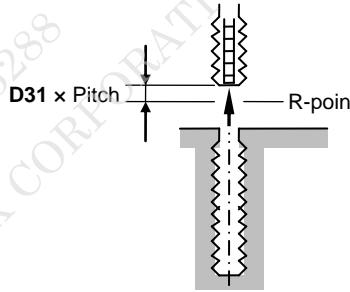
Classification	USER	Display title	POINT												
Address	Meaning	Description													
D12	Stop-hole machining hole-bottom clearance	<p>Element used to automatically set the hole-drilling depth during automatic tool development of stop-hole counter-boring and stop-hole boring units</p>  <p>Example :</p> <table border="0"> <tr> <td>SNo.</td> <td>TOOL</td> <td>NOM-ϕ</td> <td>No.</td> <td>HOLE-ϕ</td> <td>HOLE-DEP</td> </tr> <tr> <td>1</td> <td>DRILL</td> <td>10.</td> <td>10.</td> <td>(19)</td> <td></td> </tr> </table> <p style="text-align: center;">↑</p> <p style="text-align: center;">(DEPTH – tool tip compensation – D12)</p> <p>Note :</p> <p>This parameter is invalid when the residual hole diameter is not 0.</p>	SNo.	TOOL	NOM- ϕ	No.	HOLE- ϕ	HOLE-DEP	1	DRILL	10.	10.	(19)		MPL006
SNo.	TOOL	NOM- ϕ	No.	HOLE- ϕ	HOLE-DEP										
1	DRILL	10.	10.	(19)											
D13	Spot-machining hole diameter (fixed value)	<p>Hole diameter is automatically set during automatic tool development when spot-chamfering is not to be performed.</p> 	MPL007												
D14	Depth-of-cut setting element for drilling (ALMINUM)	<p>Element used to automatically set the depth-of-cut per drilling operation during automatic tool development</p> <p>HOLE-ϕ × D14 : when the material of the stock workpiece is AL (aluminum) in article MAT. 6</p> <p>HOLE-ϕ × D15 : when the material of the stock workpiece is other than AL in article MAT. 6</p>													
D15	Depth-of-cut setting element for drilling (except AL)														
	Program type	M													
	Conditions	Immediate													
	Unit	0.1													
	Setting range	0 to 10													

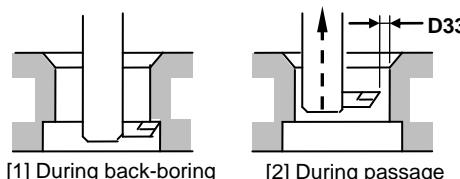
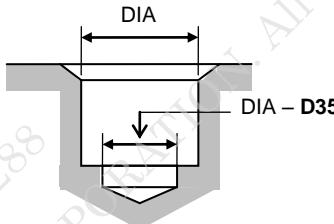
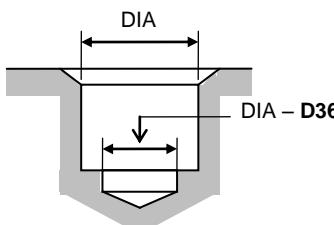
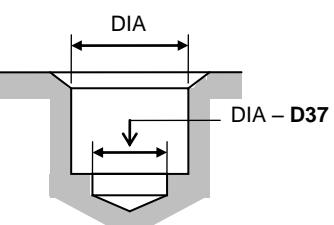
Classification	USER	Display title	POINT								
Address	Meaning	Description									
D16	<p>Number of revolutions during dwell at hole bottom for chamfering cutter or spot-machining tool in chamfering cycle</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Revolutions</td></tr> <tr><td>Setting range</td><td>0 to 9</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Revolutions	Setting range	0 to 9	<p>Z-axis feed dwell time at the hole bottom in the chamfering cycle of chamfering cutter or spot-machining tool. Set this time in spindle revolutions.</p>  <p>(Stops at hole bottom.)</p> <p>When the chamfering cutter reaches the hole bottom, the Z-axis will firstly stop moving until the spindle makes D16 revolutions, and then return to the original position at the rapid feed rate.</p> <p>Note :</p> <p>This parameter is invalid for chamfering with true-circle processing.</p>	MPL008
Program type	M										
Conditions	Immediate										
Unit	Revolutions										
Setting range	0 to 9										
D17	<p>Interference clearance of chamfering cutter</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr><td>Setting range</td><td>0 to 99</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.1 mm/0.01 in	Setting range	0 to 99	<p>The clearance in order to prevent tool interference with a wall of the workpiece or with the hole bottom during a chamfering cycle</p> <p>Use in the case of D92 bit 4 = 0: The safety clearance (D17) is radially or axially used only when an interference could actually occur.</p> <p>Note :</p> <p>A tip of the tool is allowed to enter the safety clearance area as long as there remains ever so short a clearance.</p>  <p>Use in the case of D92 bit 4 = 1: The safety clearance (D17) is radially or axially used for restrictively shifting the position of that workpiece section which is subjected to check for interference.</p> <p>Note :</p> <p>No tip of the tool is allowed to enter the safety clearance area.</p> 	MPL009
Program type	M										
Conditions	Immediate										
Unit	0.1 mm/0.01 in										
Setting range	0 to 99										

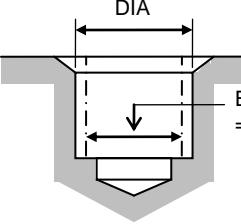
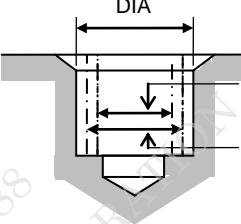
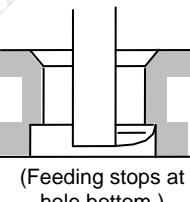
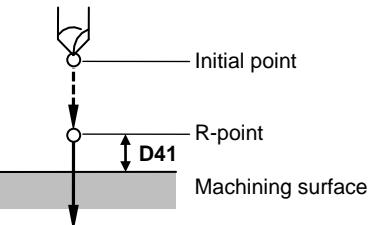
Classification	USER	Display title	POINT																		
Address	Meaning	Description																			
D18	Return feed rate for reaming or boring (cycle 3)	<p>The feed rate at which the tool is returned from the hole bottom during reaming or boring.</p>  <p>Note :</p> <ol style="list-style-type: none"> 1. Valid only when the setting of DEPTH for the reamer (tool sequence) is G01. 2. Valid only when the setting of PRE-DIA for the boring tool (tool sequence) is CYCLE 3. 3. If this parameter is 0, the tool is returned at the same feed rate as that of cutting. 	MPL010																		
D19	Number of revolutions during dwell at hole bottom for end milling	<p>Z-axis feed dwell time at the hole bottom in an end milling cycle. Set this time in spindle revolutions.</p>  <p>When the end mill reaches the hole bottom, the Z-axis will firstly stop moving until the spindle makes D19 revolutions, and then return to the original position at the rapid feed rate.</p> <p>Note :</p> <p>This parameter is invalid for true-circle processing.</p>	MPL011																		
D20	Radial depth-of-cut setting element for end milling	<p>Element used to automatically set the radial depth-of-cut per end milling operation</p> <p>Depth-of-cut = nominal diameter \times D20</p> <p>Depth-of-cut is automatically set according to the value of this parameter when nominal diameter of the end mill is input.</p> <p>Example :</p> <table border="1"> <tr> <td>SNO.</td> <td>TOOL</td> <td>NOM-Φ</td> <td>No.</td> <td>HOLE-Φ</td> <td>HOLE-DEP</td> <td>PRE-DIA</td> <td>PRE-DEP RGH</td> <td>DEPTH</td> </tr> <tr> <td>1</td> <td>END MILL</td> <td>20.</td> <td></td> <td>40.</td> <td>10.</td> <td>30.</td> <td>◆</td> <td>0. 12. (NOM-Φ \times D20) ↗</td> </tr> </table>	SNO.	TOOL	NOM-Φ	No.	HOLE-Φ	HOLE-DEP	PRE-DIA	PRE-DEP RGH	DEPTH	1	END MILL	20.		40.	10.	30.	◆	0. 12. (NOM-Φ \times D20) ↗	
SNO.	TOOL	NOM-Φ	No.	HOLE-Φ	HOLE-DEP	PRE-DIA	PRE-DEP RGH	DEPTH													
1	END MILL	20.		40.	10.	30.	◆	0. 12. (NOM-Φ \times D20) ↗													
D21	Reference bottom-finishing allowance for end milling	<p>The reference value for calculation of a bottom-finishing allowance which corresponds to the roughness level of the end milling (tool sequence). The finishing allowance in the case of roughness level 4 becomes the value of this parameter, and the values for all other roughness levels are set using the expressions listed in the table below.</p> <table border="1"> <thead> <tr> <th>Roughness</th> <th>Bottom-finishing allowance</th> </tr> </thead> <tbody> <tr> <td>0 to 3</td> <td>0.0</td> </tr> <tr> <td>4</td> <td>D21</td> </tr> <tr> <td>5</td> <td>D21 \times 0.7</td> </tr> <tr> <td>6</td> <td>D21 \times 0.7 \times 0.7</td> </tr> <tr> <td>7</td> <td>D21 \times 0.7 \times 0.7 \times 0.7</td> </tr> <tr> <td>8</td> <td>D21 \times 0.7 \times 0.7 \times 0.7 \times 0.7</td> </tr> <tr> <td>9</td> <td>D21 \times 0.7 \times 0.7 \times 0.7 \times 0.7 \times 0.7</td> </tr> </tbody> </table>	Roughness	Bottom-finishing allowance	0 to 3	0.0	4	D21	5	D21 \times 0.7	6	D21 \times 0.7 \times 0.7	7	D21 \times 0.7 \times 0.7 \times 0.7	8	D21 \times 0.7 \times 0.7 \times 0.7 \times 0.7	9	D21 \times 0.7 \times 0.7 \times 0.7 \times 0.7 \times 0.7			
Roughness	Bottom-finishing allowance																				
0 to 3	0.0																				
4	D21																				
5	D21 \times 0.7																				
6	D21 \times 0.7 \times 0.7																				
7	D21 \times 0.7 \times 0.7 \times 0.7																				
8	D21 \times 0.7 \times 0.7 \times 0.7 \times 0.7																				
9	D21 \times 0.7 \times 0.7 \times 0.7 \times 0.7 \times 0.7																				

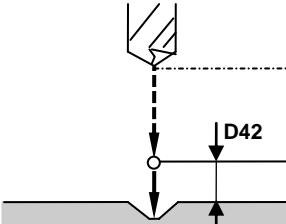
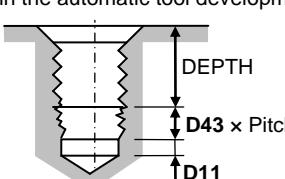
Classification	USER	Display title	POINT
Address	Meaning	Description	
D22	Tapping-cycle dwell time	Dwell time at the hole bottom or at the R-point. This value is valid when 1 is set for bit 0, 1 or 2 of parameter D91 .	
	Program type M Conditions Immediate Unit 0.01 s Setting range 0 to 99	Note : This parameter is valid only when the setting for roughness (RGH) of tapping (tool sequence) is FIX .	
D23	Prehole clearance for end milling	The excess amount of prehole diameter over nominal diameter that is used to specify whether the Z-axis is to be moved at a rapid feed rate or at a cutting feed rate during true-circle processing with the end mill	
	Program type M Conditions Immediate Unit mm/0.1 in Setting range 0 to 99	MPL012	
D24	Number of revolutions during dwell at hole bottom for boring	Z-axis feed dwell time at the hole bottom in a boring cycle. Set this time in spindle revolutions.  (Stops at hole bottom.)	When the boring bar reaches the hole bottom, the Z-axis will firstly stop moving until the spindle makes D24 revolutions, and then the spindle orientation will be performed. Note : This parameter is invalid if the roughness (RGH) of the boring (tool sequence) is 0.
	Program type M Conditions Immediate Unit Revolutions Setting range 0 to 9	MPL013	
D25	Boring-bar tip relief	The amount of relief provided for the tip of a boring bar to be kept clear of the hole wall after spindle orientation	
	Program type M Conditions Immediate Unit 0.1 mm/0.01 in Setting range 0 to 99	MPL014	Note : 1. Valid only when the setting for the prehole diameter of the boring (tool sequence) is CYCLE 1 . 2. For the relief direction of the tool tip, see the description of bit 3 and bit 4 of I14 .

Classification	USER	Display title	POINT
Address	Meaning		Description
D26	Returning distance from hole bottom for boring or back-boring		<p>The distance which the boring or back-boring tool is returned at the same feed rate and milling spindle speed as for the preceding operation after the tool has reached the hole bottom</p>  <p>[1] Has reached the hole bottom. [2] Returned at the same feedrate. [3] Returned at a rapid feedrate.</p> <p>MPL015</p>
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 in	
	Setting range	0 to 99	<p>Note : Not valid if the setting for the roughness (RGH) of the boring (tool sequence) is 1.</p>
D28	Bottom-finishing amount of boring		<p>The distance which the boring bar is fed in at 70% of the original feed rate and milling spindle speed to finish the hole bottom</p>  <p>D28</p> <p>MPL016</p>
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 in	
	Setting range	0 to 99	<p>Note : Not valid if the setting for the roughness (RGH) of the boring (tool sequence) is 1.</p>
D29	Chip removal time		<p>The time required for a chip removal tool to complete a chip removal operation after the tool has been positioned to the hole</p>
	Program type	M	
	Conditions	Immediate	
	Unit	s	
	Setting range	0 to 99	

Classification	USER	Display title	POINT												
Address	Meaning	Description													
D30	Number of incomplete threads in tapping cycle	<p>To set number of incomplete threads in tapping cycle for metric screws and unified screws. In tapping, internal thread is tapped extra for the depth of (D30 × pitch) in the direction of Z.</p> <p>This is also used as an element for automatically determining hole-drilling depth (HOLE-DEP) in the automatic tool development of the tapping unit.</p>  <p>Example :</p> <table border="1"> <thead> <tr> <th>SNO.</th> <th>TOOL</th> <th>NOM-φ</th> <th>NO.</th> <th>HOLE-φ</th> <th>HOLE-DEP</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>DRILL</td> <td>10.</td> <td></td> <td>10.</td> <td>(19)</td> </tr> </tbody> </table> <p>{DEPTH + D11 + (D30 × pitch)}↑</p> <p>[Related parameter] D43: Number of incomplete threads in tapping cycle for piped screws</p>	SNO.	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP	1	DRILL	10.		10.	(19)	MPL017
SNO.	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP										
1	DRILL	10.		10.	(19)										
D31	Tapper elongation amount for tapping	<p>Excess amount of tool return due to elongation of the tapper during tapping cycle</p> <p>Set this value in spindle revolutions.</p> 	MPL018												
D32	Number of spindle revolutions until spindle CCW rotation begins in tapping cycle	The number of inertial turns in tapping cycle that the spindle has rotated clockwise during the time from output of a spindle CCW rotation command to the start of spindle CCW rotation													

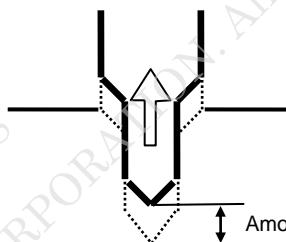
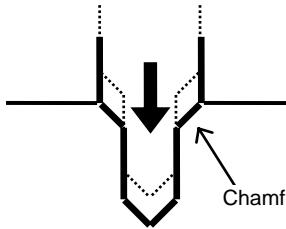
Classification	USER	Display title	POINT
Address		Meaning	
D33		<p>Back-boring tool tip relief</p> <p>The amount of relief provided for a back-boring tool tip to be kept clear of the prehole walls as it is being passed through the prehole in the oriented state of the spindle</p> 	
		Program type	M
		Conditions	Immediate
		Unit	0.1 mm/0.01 in
		Setting range	0 to 99
D35		<p>Prehole-drilling diameter setting element for reamer (drilling)</p> <p>Element used to automatically set the prehole-drilling diameter during automatic tool development of the reamer unit (When the pre-machining process is drilling.)</p> 	
		Program type	M
		Conditions	Immediate
		Unit	0.01 mm/0.001 in
		Setting range	0 to 999
D36		<p>Prehole-drilling diameter setting element for reamer (boring)</p> <p>Element used to automatically set the prehole-drilling diameter during automatic tool development of the reamer unit (When the pre-machining process is boring.)</p> 	
		Program type	M
		Conditions	Immediate
		Unit	0.01 mm/0.001 in
		Setting range	0 to 999
D37		<p>Prehole-drilling diameter setting element for reamer (end milling)</p> <p>Element used to automatically set the prehole-drilling diameter during automatic tool development of the reamer unit (When the pre-machining process is end milling.)</p> 	
		Program type	M
		Conditions	Immediate
		Unit	0.01 mm/0.001 in
		Setting range	0 to 999

Classification	USER	Display title	POINT
D38	Reamer-prehole diameter setting element for boring or end milling	<p>1. In automatic tool development of the reamer unit, if the pre-machining process is boring:</p>  <p>MPL021</p>	
	Program type M		
	Conditions Immediate		
	Unit 0.01 mm/0.001 in		
	Setting range 0 to 999		
D39	Reamer-prehole diameter setting element for end milling	<p>2. In automatic tool development of the reamer unit, if the pre-machining process is end milling:</p>  <p>MPL022</p>	
	Program type M		
	Conditions Immediate		
	Unit 0.01 mm/0.001 in		
	Setting range 0 to 999		
D40	Number of revolutions during dwell at spot-faced hole bottom for inverted spot-facing	<p>Z-axis feed dwell time at the spot-faced hole bottom in an inversed spot facing cycle. Set this time in spindle revolutions.</p>  <p>When the inverted spot-facing tool reaches the hole bottom, firstly the Z-axis will stop moving until the spindle makes D40 revolutions, and then the rotational direction of the spindle will reverse.</p> <p>MPL023</p>	
	Program type M		
	Conditions Immediate		
	Unit Revolutions		
	Setting range 0 to 9		
D41	R-point height during point-machining	<p>R-point height of each tool in the point-machining unit</p> <p>Example :</p>  <p>MPL024</p>	
	Program type M		
	Conditions Immediate		
	Unit mm/0.1 in		
	Setting range 0 to 99	(⇒D1, D42)	

Classification	USER	Display title	POINT																
Address	Meaning	Description																	
D42	<p>Height of the third R-point during point machining</p> <table border="1" style="margin-left: 10px;"> <tr> <td>Program type</td><td>M</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr> <td>Setting range</td><td>-999 to 999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.1 mm/0.01 in	Setting range	-999 to 999	<p>Height of the third R-point</p>  <p>The height of the R-point during point machining is basically D41, however it is changed to D42 under the following conditions.</p> <table border="1" style="margin-left: 10px;"> <thead> <tr> <th>Tool sequence</th><th>Conditions</th></tr> </thead> <tbody> <tr> <td>Drill</td><td> <ul style="list-style-type: none"> Bit 6 of parameter D91 is set to 1 (D42 valid). There is a drill in the pre-machining tool sequence of the same unit. </td></tr> <tr> <td>Chamfering cutter</td><td> <ul style="list-style-type: none"> Bit 7 of parameter D91 is set to 1 (D42 valid). CYCLE 2 is selected for the machining cycle. </td></tr> <tr> <td>Spot</td><td> <ul style="list-style-type: none"> Bit 7 of parameter D91 is set to 1 (D42 valid). CYCLE 2 in the chamfering cycle is selected for the machining cycle. </td></tr> </tbody> </table>	Tool sequence	Conditions	Drill	<ul style="list-style-type: none"> Bit 6 of parameter D91 is set to 1 (D42 valid). There is a drill in the pre-machining tool sequence of the same unit. 	Chamfering cutter	<ul style="list-style-type: none"> Bit 7 of parameter D91 is set to 1 (D42 valid). CYCLE 2 is selected for the machining cycle. 	Spot	<ul style="list-style-type: none"> Bit 7 of parameter D91 is set to 1 (D42 valid). CYCLE 2 in the chamfering cycle is selected for the machining cycle. 	MPL001
Program type	M																		
Conditions	Immediate																		
Unit	0.1 mm/0.01 in																		
Setting range	-999 to 999																		
Tool sequence	Conditions																		
Drill	<ul style="list-style-type: none"> Bit 6 of parameter D91 is set to 1 (D42 valid). There is a drill in the pre-machining tool sequence of the same unit. 																		
Chamfering cutter	<ul style="list-style-type: none"> Bit 7 of parameter D91 is set to 1 (D42 valid). CYCLE 2 is selected for the machining cycle. 																		
Spot	<ul style="list-style-type: none"> Bit 7 of parameter D91 is set to 1 (D42 valid). CYCLE 2 in the chamfering cycle is selected for the machining cycle. 																		
D43	<p>Number of incomplete threads in tapping cycle for piped screw</p> <table border="1" style="margin-left: 10px;"> <tr> <td>Program type</td><td>M</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>Thread</td></tr> <tr> <td>Setting range</td><td>0 to 9</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Thread	Setting range	0 to 9	<p>To set number of incomplete threads in tapping cycle for piped screws (PT, PF, PS). In tapping, internal thread is tapped extra for the depth of (D43 × pitch) in the direction of Z.</p> <p>This is also used as an element for automatically determining hole-drilling depth (HOLE-DEP) in the automatic tool development of the tapping unit.</p> 	MPL017								
Program type	M																		
Conditions	Immediate																		
Unit	Thread																		
Setting range	0 to 9																		
D44	<p>Selecting an automatic calculation method for the amount of chamfer with the tapping unit</p> <table border="1" style="margin-left: 10px;"> <tr> <td>Program type</td><td>M</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>—</td></tr> <tr> <td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	—	Setting range	0, 1	<p>Select an automatic calculation method for the amount of chamfer with the tapping unit.</p> <p>0: Following calculation expression is used:</p> $\text{Amount of chamfer} = \frac{(\text{MAJOR-}\phi + 2 \times \text{PITCH}) - \text{PRE-DIA}}{2}$ <p>1: Following calculation expression is used (same as M32):</p> $\text{Amount of chamfer} = \frac{\text{MAJOR-}\phi - \text{PRE-DIA}}{2}$ <p>Note :</p> <p>Select 1 if a loss of the threaded section is likely due to overchamfering as compared with a small TAP-DEP value.</p> <p>Example : Sheet metal tapping</p>									
Program type	M																		
Conditions	Immediate																		
Unit	—																		
Setting range	0, 1																		

Classification	USER	Display title	POINT												
D45	Gradual decrements in drilling depth		<p>Description</p> <p>Drilling depth</p> <p>Number of times</p> <p>q₁: 1st drilling depth q_i: i-th drilling depth q_n: Residual hole depth b : Minimum drilling depth D46 $q_i = q_1 - D45 \times (i - 1)$ (If $q_i \geq b$) $q_i = b$ (If $q_i < b$)</p> <p>(⇒D46)</p>												
D46	Minimum gradual drilling depth		<p>Set the minimum gradual drilling depth. However, if the residual hole depth is smaller than D46, actual drilling depth will be the same as the residual hole depth.</p> <p>(⇒D45)</p>												
D47	Reamer-prehole machining overshoot		<p>Element used to automatically set the hole depth (HOLE-DEP) of drilling, end milling and boring during automatic tool development of the reamer unit</p> <p>For drilling For end milling or boring</p> <p>MPL025</p> <p>Example :</p> <table border="1"> <tr> <td>SNO.</td> <td>TOOL</td> <td>NOM-φ</td> <td>No.</td> <td>HOLE-φ</td> <td>HOLE-DEP</td> </tr> <tr> <td>1</td> <td>DRILL</td> <td>10.</td> <td>10.</td> <td>21.</td> <td>← (DEPTH + D47)</td> </tr> </table>	SNO.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	1	DRILL	10.	10.	21.	← (DEPTH + D47)
SNO.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP										
1	DRILL	10.	10.	21.	← (DEPTH + D47)										

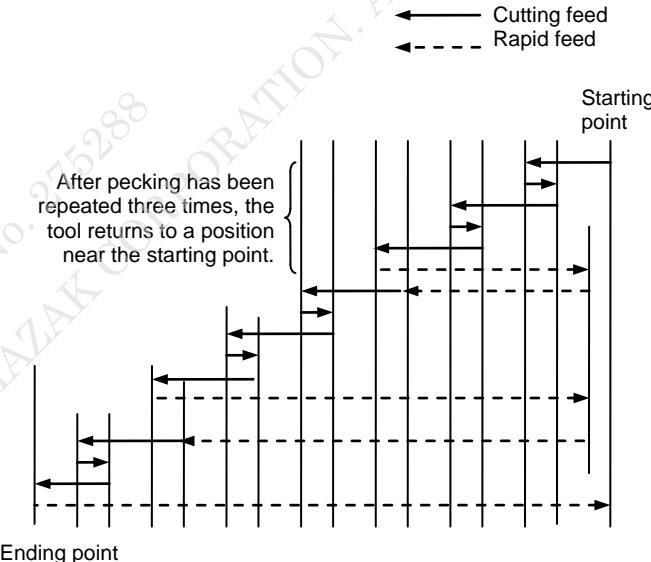
Classification	USER	Display title	POINT
Address		Meaning	
D48		Feed override for the section to be chamfered in the planetary tapping cycle Feed override for the section to be chamfered in the planetary tapping cycle	
		Program type	M
		Conditions	Immediate
		Unit	%
		Setting range	0 to 999
D49		Amount of return at hole bottom during the planetary tapping cycle Amount of return at hole bottom during the planetary tapping cycle	
		Program type	M
		Conditions	Immediate
		Unit	0.1 thread
		Setting range	0 to 999
D50		Auto-set feed rate for pre-hole machining in the planetary tapping cycle The feed rate for pre-hole machining will be auto-set to D50 when the planetary tapping cycle is selected.	
		SNo.	TOOL NOM- ϕ NO.HOLE- ϕ HOLE-DEP PRE-DIA PRE-DEP RGH DEPTH C-SP FR M M
		1	TAP M10. 10. 23.7 PLANET 0.15 FIX P1.5 50 1.5
			D50
		Program type	M
D51		Auto-set feed rate for planetary tapping cycle The feed rate will be auto-set to D51 when the planetary tapping cycle is selected.	
		SNo.	TOOL NOM- ϕ NO.HOLE- ϕ HOLE-DEP PRE-DIA PRE-DEP RGH DEPTH C-SP FR M M
		1	TAP M10. 10. 23.7 PLANET 0.15 FIX P1.5 50 0.04
			D51
		Program type	M
		Conditions Immediate	
		Unit 0.01 mm/rev 0.001 in/rev	
		Setting range 0 to 9999	

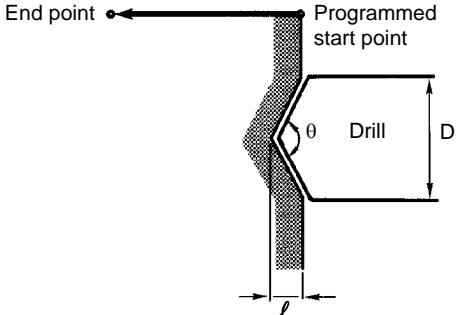
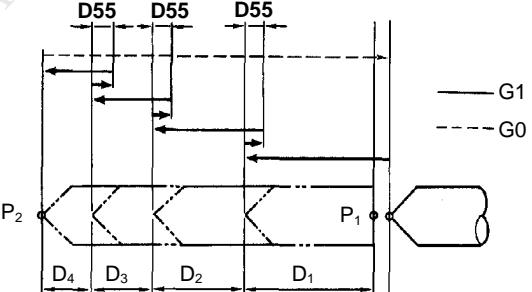


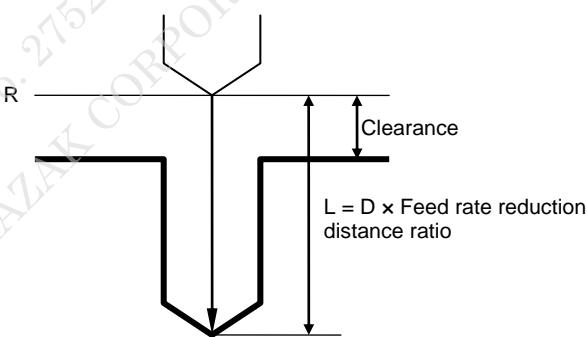
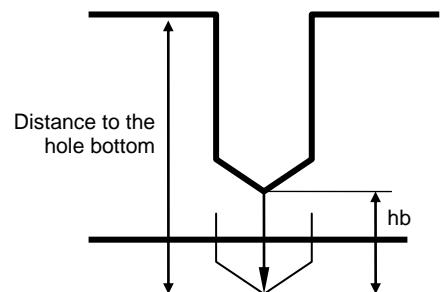
The amount of return at hole bottom during the planetary tapping cycle
Specify data by the number of threads.
Amount of return = Tapping pitch × **D49/10**

SNo. TOOL NOM- ϕ NO.HOLE- ϕ HOLE-DEP PRE-DIA PRE-DEP RGH DEPTH C-SP FR M M
1 TAP M10. 10. 23.7 PLANET 0.15 FIX P1.5 50 1.5
D50

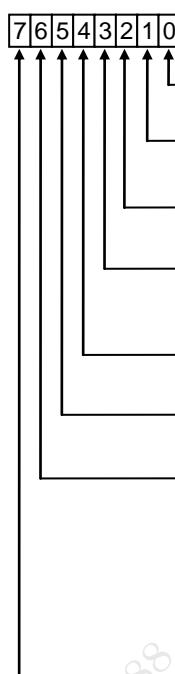
The feed rate will be auto-set to **D51** when the planetary tapping cycle is selected.
SNo. TOOL NOM- ϕ NO.HOLE- ϕ HOLE-DEP PRE-DIA PRE-DEP RGH DEPTH C-SP FR M M
1 TAP M10. 10. 23.7 PLANET 0.15 FIX P1.5 50 0.04
D51

Classification	USER	Display title	POINT
Address	Meaning	Description	
D52	Reduction ratio for the G00-based relief rate during a very-deep-hole drilling cycle	Set the reduction ratio for the G00-based relief rate during the very-deep-hole drilling in a drilling or turning-drilling unit.	
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
D53	Number of times of pecking up to the return of the tool to a position near the starting point of the very-deep-hole drilling cycle of a drilling or turning-drilling unit	Set the number of times of pecking to be executed before returning the tool to a position near the starting point of the very-deep-hole drilling cycle of a drilling or turning-drilling unit. Example : If D53 = 3:	
	Program type	M	
	Conditions	Immediate	
	Unit	Times	
	Setting range	0 to 9999	

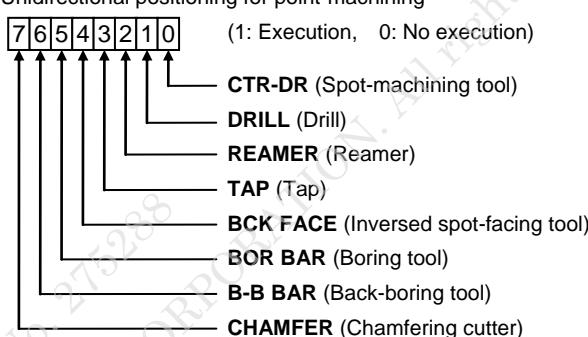
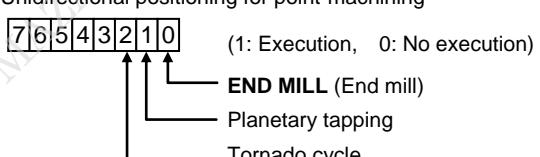
Classification	USER	Display title	POINT
Address	Meaning	Description	
D54	<p>Deceleration rate at cutting start for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle</p> <p>Program type M</p> <p>Conditions Immediate</p> <p>Unit %</p> <p>Setting range 0 to 100</p>	<p>$F' = F \times \frac{D54}{100}$</p> <p>F : Specified feed rate F' : Feed rate at cutting start</p>  <p>Feed rate at F' over distance ℓ from the programmed start point</p> $\begin{cases} \ell = \frac{D}{2 \times \tan\theta/2} & (0^\circ < \theta < 180^\circ) \\ \ell = 0 & (\theta \geq 180^\circ) \end{cases}$ <p>NM211-00268</p>	
D55	<p>Drilling return distance for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle</p> <p>Program type M</p> <p>Conditions Immediate</p> <p>Unit 0.001 mm/0.0001 in</p> <p>Setting range 0 to 9999</p>	 <p>P₁: Start point P₂: End point D_n ($n = 1$ to 4) = Cut depth</p> <p>NM211-00252</p>	
D56	<p>Number of revolutions during dwell at chip ejection position and hole bottom for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle</p> <p>Program type M</p> <p>Conditions Immediate</p> <p>Unit Revolutions</p> <p>Setting range 0 to 255</p>	<p>Set the number of revolutions of the milling spindle during dwell time at the chip ejection position and the hole bottom for the very-deep-hole drilling cycle or the decremental very-deep-hole machining cycle.</p>	

Classification	USER	Display title	POINT
Address	Meaning		Description
D57	Return speed for very-deep-hole drilling cycle/decremental very-deep-hole machining cycle	Program type Conditions Unit Setting range	Set the return speed for the very-deep-hole drilling cycle or the decremental very-deep-hole machining cycle. M Immediate 0.001 mm/rev 0.0001 in/rev 0 to 9999
D58	Feed rate reduction distance ratio at cutting start of a very-deep-hole drilling cycle (blind hole, through hole)	Program type Conditions Unit Setting range	Set the feed rate reduction distance from the reference point at cutting start of a very-deep-hole drilling cycle (blind hole, through hole) during the point machining. Specify the ratio with respect to the drill diameter. If the nominal diameter of the drill is D, the feed rate reduction distance L at cutting start is calculated using the following calculation expression: $L = D \times D58/100$ The feed rate is reduced by the ratio specified in D54 (feed rate reduction ratio at cutting start) by the distance L from the reference point (R). 
D59	Surface speed reduction ratio at cutting end of a very-deep-hole drilling cycle (through hole)	Program type Conditions Unit Setting range	Set the surface speed reduction ratio at cutting end of a very-deep-hole drilling cycle (through hole) during the point machining. If the surface speed during drilling is S, the surface speed S' at cutting end is calculated using the following calculation expression: $S' = S \times D59/100$ The surface speed is reduced to S' by "speed reduction distance immediately before the hole bottom" (hb specified in the program). 

Classification	USER	Display title	POINT
Address	Meaning		Description
D60	Automatic setting ratio of axial cutting feed rate during chamfering		Set the ratio between automatic setting value for the feed rate in the axial cutting and that in the radial cutting during chamfering in the point machining.
	Program type	M	
	Conditions	After movement stop	
	Unit	%	
	Setting range	0 to 100	
D61	Nominal diameter of chamfering cutter, set during automatic tool development for a point machining unit		Set the nominal diameter of a chamfering cutter that is automatically set during automatic tool development for a point machining unit of MAZATROL programs.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 in	
	Setting range	0 to 999	
D73 to D77	Learning of cutting conditions (DEP-Z range)		Specify DEP-Z range for the end mill and the face mill from the learning data of cutting conditions. When learning data on the condition that DEP-Z is in the following range has been stored in the memory, learning is not effectuated again. For a DEP-Z range of the end mill, set a value of "DEPTH/NOM-Φ" (at a unit of 0.1%). 0 to D73 DEP-Z range (for end mill) 1 D73 to D74 DEP-Z range (for end mill) 2 D74 to D75 DEP-Z range (for end mill) 3
	Program type	M	
	Conditions	Immediate	
	Unit	0.1% (0.1 mm/0.01 in)	For a DEP-Z range of the face mill, set a value of "DEPTH" (at a unit of 0.1 mm or 0.01 in). 0 to D76 DEP-Z range (for face mill) 1 D76 to D77 DEP-Z range (for face mill) 2
	Setting range	0 to 1000	
D78 to D82	Learning of cutting conditions (WID-R range)		Specify WID-R range for the boring bar, back boring bar and end mill from the learning data of cutting conditions. When learning data on the condition that WID-R is in the following range has been stored in the memory, learning is not effectuated again. For a WID-R range of the boring bar and back boring bar, set a value of "DEPTH" (at a unit of 0.1 mm/0.01 in). 0 to D78 WID-R range (for boring bar and back boring bar) 1 D78 to D79 WID-R range (for boring bar and back boring bar) 2
	Program type	M	
	Conditions	Immediate	
	Unit	0.1% (0.1 mm/0.01 in)	For a WID-R range of the end mill, set a value of "DEPTH/NOM-Φ" (at a unit of 0.1%). 0 to D80 WID-R range (for end mill) 1 D80 to D81 WID-R range (for end mill) 2 D81 to D82 WID-R range (for end mill) 3
	Setting range	0 to 1000	

Classification	USER	Display title	POINT
D91	—		<p>Description</p>  <ul style="list-style-type: none"> (1: Execution, 0: No execution) M04 is output after the tool has dwelled at the hole bottom during a tapping cycle. The tool dwells after M04 has been output at the hole bottom during a tapping cycle. The tool dwells after it has been returned to the R-point during a tapping cycle. If a drill is used in the pre-machining of the centering drill cycle, the R-point height is set to D1. The finishing tool path is shortened during a true-circle processing cycle (end milling). The tool path is shortened during a true-circle processing cycle (chamfering). If a pre-machining tool sequence is included in the same unit, the R-point height of the drill is set to D1 or D42. 1. The R-point height of the chamfering cutter during the cycle 2 is set to D42. 2. The R-point height of the spot-machining tool during the chamfering cycle (cycle 2) is set to D42. 3. R-point height selection for smooth chamfering (point machining) <ul style="list-style-type: none"> 0 : The R-point height is set to TC39. 1 : The R-point height is set to D42.
D92 (bit 0)	During a true-circle processing (end milling) cycle, E17 is used/not used for axial feed		<p>1: Used 0: Not used</p>
D92 (bit 1)	The R1-point height of the back spot facing is set/not set to D1		<p>1: Set 0: Not set</p>

Classification	USER	Display title	POINT
Address	Meaning		Description
D92 (bit 2)	If a chamfering cutter is included in the pre-machining tool sequence of the same unit, the R-point height of the reamer is set/not set to D1		1: Set 0: Not set
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
	If a chamfering cutter is included in the pre-machining tool sequence of the same unit, the R-point height of the tap is set/not set to D1		1: Set 0: Not set
D92 (bit 3)	The position cleared radially or axially by D17 from the workpiece section concerned is subjected/not subjected to check for interference		1: Subjected 0: Not subjected
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
	Dwell time designation for synchronous tapping in the MAZATROL program		1: Valid 0: Invalid Note : Timing of dwell execution of bit 5 is determined by settings in bit 0 to 2 of parameter D91 .
D92 (bit 5)	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	

Classification	USER	Display title	POINT
Address	Meaning		Description
D92 (bit 6)	During planetary tapping, chips are ejected/not ejected automatically prior to the threading process		1: Ejected 0: Not ejected
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
D93	—		Unidirectional positioning for point-machining 
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
D94	—		Unidirectional positioning for point-machining 
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

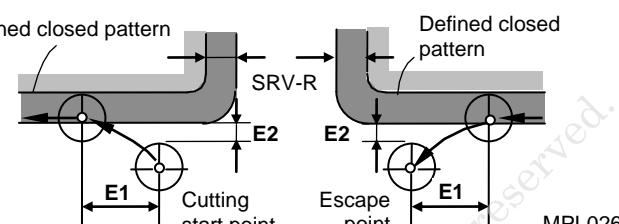
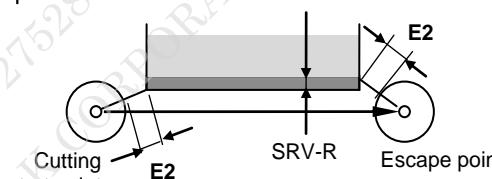
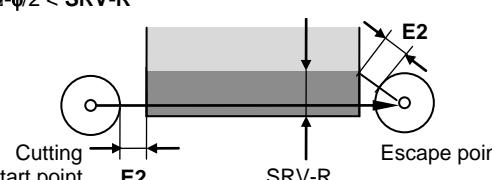
Classification	USER	Display title	POINT								
Address	Meaning	Description									
D95		<p>Selection of the auto-setting method to be used for the MAZATROL program data items of the tapping unit (TAPPING and CBOR-TAP) and the diameter item of pipe taps on the TOOL DATA display.</p> <table border="1"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>Binary, eight digits</td> </tr> </table>		Program type	M	Conditions	Immediate	Unit	Bit	Setting range	Binary, eight digits
Program type	M										
Conditions	Immediate										
Unit	Bit										
Setting range	Binary, eight digits										
D96 (bit 0)		<p>Surface speed calculation for spot drill used</p> <p>0: Based on machining hole diameter 1: Based on nominal diameter of the tool</p> <table border="1"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>0, 1</td> </tr> </table>		Program type	M	Conditions	Immediate	Unit	Bit	Setting range	0, 1
Program type	M										
Conditions	Immediate										
Unit	Bit										
Setting range	0, 1										
D96 (bit 1)		<p>Element that determines the counterboring pattern (whether to execute diameter enlarging)</p> <p>0: Determines the counterboring pattern from the nominal diameter in tool data and the hole diameter in the program. 1: Determines the counterboring pattern from the tool diameter in the tool data and the hole diameter in the program.</p> <table border="1"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>After stop of movement</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>0, 1</td> </tr> </table>		Program type	M	Conditions	After stop of movement	Unit	Bit	Setting range	0, 1
Program type	M										
Conditions	After stop of movement										
Unit	Bit										
Setting range	0, 1										

Classification	USER	Display title	POINT								
Address	Meaning		Description								
D96 (bit 3)	<p>Menu display setting of RGH BCB (inversed faced hole machining), BK-CBORE (back boring), and CBOR-TAP (counterbore tapping), as well as BACKSPOT FACER (inversed spot-facing tool) and BACK BORING BAR (back boring) in the tool sequence data</p> <table border="1"> <tr> <td>Program type</td><td>M</td></tr> <tr> <td>Conditions</td><td>At power on</td></tr> <tr> <td>Unit</td><td>Bit</td></tr> <tr> <td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	At power on	Unit	Bit	Setting range	0, 1		<p>RGH BCB (inversed faced hole machining), BK-CBORE (back boring), and CBOR-TAP (counterbore tapping) are displayed in the menu in the point-machining unit.</p> <p>BACKSPOT FACER (inversed spot-facing tool) and BACK BORING BAR (back boring) are displayed in the tool menu for tool sequence data in the point-machining unit.</p> <p>0 : Invalid 1 : Valid</p>
Program type	M										
Conditions	At power on										
Unit	Bit										
Setting range	0, 1										
D97 (bit 0)	<p>Cutting drawing method for circular milling unit on the VIRTUAL MACHINING display</p> <table border="1"> <tr> <td>Program type</td><td>M</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>Bit</td></tr> <tr> <td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Bit	Setting range	0, 1		<p>Select the cutting drawing method for circular milling unit on the VIRTUAL MACHINING display.</p> <p>0: Detailed drawing (Cutting is drawn according to the specified pitch.)</p> <p>1: Simple drawing (Cutting is drawn using approximate geometries regardless of the specified pitch. Cutting is performed from the start point to the end point by one single rotation and another rotation at the end point.)</p>
Program type	M										
Conditions	Immediate										
Unit	Bit										
Setting range	0, 1										
D105 (bit 0)	<p>Handling the X-axis coordinates of non-turning TPC relay points</p> <table border="1"> <tr> <td>Program type</td><td>M</td></tr> <tr> <td>Conditions</td><td>After stop of movement</td></tr> <tr> <td>Unit</td><td>Bit</td></tr> <tr> <td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	After stop of movement	Unit	Bit	Setting range	0, 1		<p>0: Radius value 1: Diameter value</p>
Program type	M										
Conditions	After stop of movement										
Unit	Bit										
Setting range	0, 1										
D105 (bit 1)	<p>Tap high-speed deep-hole cycle menu display/hidden</p> <table border="1"> <tr> <td>Program type</td><td>M</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>—</td></tr> <tr> <td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	—	Setting range	0, 1		<p>Tap high-speed deep-hole cycle menu</p> <p>0: Hidden 1: Display</p>
Program type	M										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										

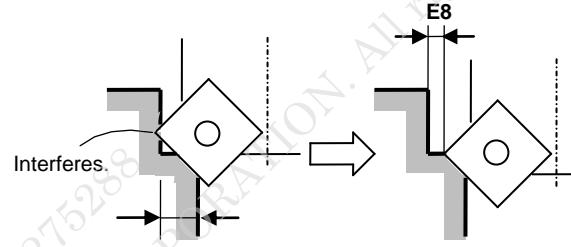
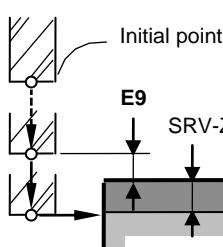
Classification	USER	Display title	POINT								
Address	Meaning		Description								
D106 (bit 0)	<p>Machine operation when the first machining unit is not preceded by an index unit</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Bit</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>		Program type	M	Conditions	Immediate	Unit	Bit	Setting range	0, 1	<p>Select whether or not to generate alarm 1602 SET INDEX UNIT BEFORE MACHINING when the following unit is not preceded by an index unit.</p> <p>[Units]</p> <p>Point machining, line machining, chamfering, face machining, C-axis point machining, C-axis line machining, C-axis chamfering, turning, EIA/ISO subprogram, manual program, MMS, pallet change, workpiece measurement, tool measurement</p> <p>0: No alarm 1: Alarm</p>
Program type	M										
Conditions	Immediate										
Unit	Bit										
Setting range	0, 1										
D106 (bit 1)	<p>Execution of an index command after the second or subsequent work coordinate command in WPC unit</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Bit</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>		Program type	M	Conditions	Immediate	Unit	Bit	Setting range	0, 1	<p>0: Do not execute 1: Execute</p>
Program type	M										
Conditions	Immediate										
Unit	Bit										
Setting range	0, 1										
D106 (bit 2)	<p>T-code output during indexing before the end unit</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Bit</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>		Program type	M	Conditions	Immediate	Unit	Bit	Setting range	0, 1	<p>T-code in index unit to be executed before the end unit</p> <p>0: Output 1: Do not output</p>
Program type	M										
Conditions	Immediate										
Unit	Bit										
Setting range	0, 1										
D106 (bit 4)	<p>Selection of rotational axis command system by table selection</p> <p>Selection of rotational axis command system by table selection</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Bit</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>		Program type	M	Conditions	Immediate	Unit	Bit	Setting range	0, 1	<p>Selection of rotational axis command system by table selection</p> <p>0: Table selection invalid 1: Table selection valid Specify the rotational axis in the table selection unit when table selection is valid.</p>
Program type	M										
Conditions	Immediate										
Unit	Bit										
Setting range	0, 1										

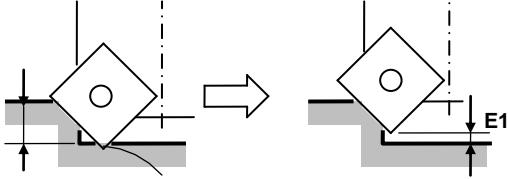
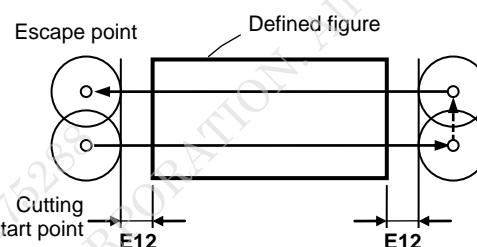
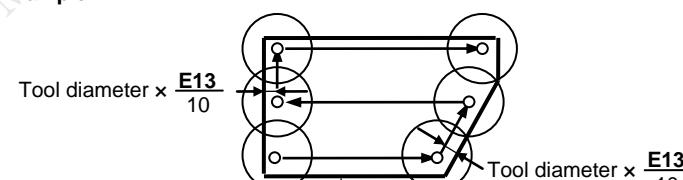
Classification	USER	Display title	POINT
Address	Meaning		Description
D106 (bit 5)	Alarm output timing when the first machining unit is not preceded by an index unit		Timing of outputting alarm 1602 SET INDEX UNIT BEFORE MACHINING when bit 0 of parameter D106 is set to 1 0: Output before executing a machining unit 1: Alarm output immediately after the cycle start button is pressed
Program type	M		
Conditions	Immediate		
Unit	Bit		
Setting range	0, 1		

2-3-3 User parameter LINE/FACE/3D (E)

Classification	USER	Display title	LINE/FACE/3D
Address	Meaning		Description
E1	Closed-pattern cutting start point and escape point setting element		<p>Element used to set cutting start point and escape point for closed-pattern line- or face-machining</p> <p>Example :</p>  <p>MPL026</p>
	Program type	M	[Applicable units]
	Conditions	Immediate	<ul style="list-style-type: none"> LINE OUT, LINE IN, CHMF OUT and CHMF IN Wall finishing of STEP, POCKET, PCKT MT and PCKT VLY
	Unit	0.1 mm/0.01 in	
	Setting range	0 to 999	
E2	Cutting start point and escape point setting element (the first clearance)		<p>Element used to set the cutting start point and escape point for line- or face-machining (the first clearance)</p> <p>Example :</p> <p>$\text{NOM-}\phi/2 \geq \text{SRV-R}$</p>  <p>$\text{NOM-}\phi/2 < \text{SRV-R}$</p>  <p>MPL027</p>
	Program type	M	[Applicable units]
	Conditions	Immediate	<ul style="list-style-type: none"> All line-machining units Face-machining units other than FCE MILL, TOP EMIL, and SLOT
	Unit	0.1 mm/0.01 in	Note :
	Setting range	0 to 999	<ol style="list-style-type: none"> See the diagram of parameter E1 also. Positioning of E2 at the escape point can be selected using E95, but only for line-machining units.

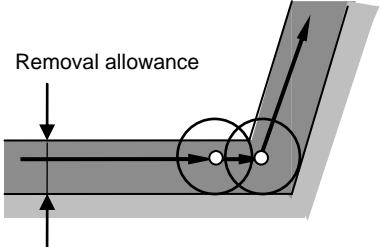
Classification	USER	Display title	LINE/FACE/3D															
Address	Meaning	Description																
E4	Reference allowance of finishing in radial direction		The reference value of each finishing allowance R (FIN-R) which is automatically set when the roughness levels of the line- or face-machining units have been set The finishing allowance R in the case of roughness level 4 becomes the value of this parameter, and the values for all other roughness levels are calculated using the expressions listed in the table below.															
	Program type	M	<table border="1"> <thead> <tr> <th>Roughness</th><th>FIN-R</th></tr> </thead> <tbody> <tr><td>0 to 3</td><td>0.0</td></tr> <tr><td>4</td><td>E4</td></tr> <tr><td>5</td><td>E4 × 0.7</td></tr> <tr><td>6</td><td>E4 × 0.7 × 0.7</td></tr> <tr><td>7</td><td>E4 × 0.7 × 0.7 × 0.7</td></tr> <tr><td>8</td><td>E4 × 0.7 × 0.7 × 0.7 × 0.7</td></tr> <tr><td>9</td><td>E4 × 0.7 × 0.7 × 0.7 × 0.7 × 0.7</td></tr> </tbody> </table>	Roughness	FIN-R	0 to 3	0.0	4	E4	5	E4 × 0.7	6	E4 × 0.7 × 0.7	7	E4 × 0.7 × 0.7 × 0.7	8	E4 × 0.7 × 0.7 × 0.7 × 0.7	9
Roughness	FIN-R																	
0 to 3	0.0																	
4	E4																	
5	E4 × 0.7																	
6	E4 × 0.7 × 0.7																	
7	E4 × 0.7 × 0.7 × 0.7																	
8	E4 × 0.7 × 0.7 × 0.7 × 0.7																	
9	E4 × 0.7 × 0.7 × 0.7 × 0.7 × 0.7																	
Conditions	Immediate																	
Unit	0.1 mm/0.01 in																	
Setting range	0 to 999																	
E5	Element used to set the cutting start point and escape point (the second clearance)		Element used to set the cutting start point and escape point (the second clearance) E2 is used generally as a clearance on the X-Y plane, however, E5 is used when the condition meets both of 1. and 2. mentioned below. 1. There is pre-machining in the same unit. 2. The parameter (E91 to E95) that makes E5 effective is set to ON (1). [Applicable units] LINE OUT, LINE IN, STEP, POCKET, POCKT MT, PCKT VLY [Related parameters] E91 bit 3, E92 bit 3, E93 bit 3, E94 bit 3, E95 bit 7 * Parameter that effectuates E5 in the applicable unit.															
	Program type	M																
	Conditions	Immediate																
	Unit	0.1 mm/0.01 in																
	Setting range	0 to 999																
E6	Reference allowance of finishing in axial direction		The reference value of each finishing allowance Z (FIN-Z) which is automatically set when the roughness levels of the line- or face-machining units have been set The finishing allowance Z in the case of roughness level 4 becomes the value of this parameter, and the values for all other roughness levels are calculated using the expressions listed in the table below.															
	Program type	M	<table border="1"> <thead> <tr> <th>Roughness</th><th>FIN-Z</th></tr> </thead> <tbody> <tr><td>0 to 3</td><td>0.0</td></tr> <tr><td>4</td><td>E6</td></tr> <tr><td>5</td><td>E6 × 0.7</td></tr> <tr><td>6</td><td>E6 × 0.7 × 0.7</td></tr> <tr><td>7</td><td>E6 × 0.7 × 0.7 × 0.7</td></tr> <tr><td>8</td><td>E6 × 0.7 × 0.7 × 0.7 × 0.7</td></tr> <tr><td>9</td><td>E6 × 0.7 × 0.7 × 0.7 × 0.7 × 0.7</td></tr> </tbody> </table>	Roughness	FIN-Z	0 to 3	0.0	4	E6	5	E6 × 0.7	6	E6 × 0.7 × 0.7	7	E6 × 0.7 × 0.7 × 0.7	8	E6 × 0.7 × 0.7 × 0.7 × 0.7	9
Roughness	FIN-Z																	
0 to 3	0.0																	
4	E6																	
5	E6 × 0.7																	
6	E6 × 0.7 × 0.7																	
7	E6 × 0.7 × 0.7 × 0.7																	
8	E6 × 0.7 × 0.7 × 0.7 × 0.7																	
9	E6 × 0.7 × 0.7 × 0.7 × 0.7 × 0.7																	
Conditions	Immediate																	
Unit	0.1 mm/0.01 in																	
Setting range	0 to 999																	

Classification	USER	Display title	LINE/FACE/3D																				
Address	Meaning	Description																					
E7	Allowance of cutting start point in axial direction (the second clearance)		Allowance of cutting start point in axial direction For the line- or face-machining, E9 is used as an axial clearance for rapid access to the machining point from the initial point, however, E7 is used when the condition meets both of 1. and 2. mentioned below. 1. There is pre-machining in the same unit. 2. The parameter (E91 to E97) that makes E7 effective is set to ON (1). [Applicable units] All line-/face-machining units except the face milling and angular face unit. [Related parameters] E91 bit 2, E92 bit 2, E93 bit 2, E94 bit 2, E95 bit 6, E96 bit 1, E97 bit 2 * Parameter that effectuates E7 in the applicable unit.																				
	Program type	M																					
	Conditions	Immediate																					
	Unit	0.1 mm/0.01 in																					
	Setting range	0 to 99																					
E8	Radial interference clearance of chamfering cutter		The amount of clearance that prevents interference of the chamfering cutter with the walls during face-machining  MPL028																				
	Program type	M																					
	Conditions	Immediate																					
	Unit	0.1 mm/0.01 in																					
	Setting range	0 to 999																					
E9	Allowance of axial-cutting start position (the first clearance)		Element used to set the position in which the cutting feed in axial direction is to be started after the line- or face-machining tool has been moved from the initial point toward the workpiece at a rapid feed rate Example :  MPL029																				
	Program type	M																					
	Conditions	Immediate																					
	Unit	0.1 mm/0.01 in																					
	Setting range	0 to 999																					
E10	Depth-of-cut-R automatic setting element (Face milling, End milling-top, End milling-step)		Element used to automatically set the radial depth-of-cut (WID-R) of the tool sequence in FCE MILL, TOP EMIL or STEP unit $\text{WID-R} = \frac{\text{NOM-}\phi \times \text{E10}}{10}$ Example : <table border="0"><tr><td>SNo.</td><td>TOOL</td><td>NOM-ϕ</td><td>No.</td><td>APRCH-X</td><td>APRCH-Y</td><td>TYPE</td><td>ZFD</td><td>DEP-Z</td><td>WID-R</td></tr><tr><td>R1</td><td>FCE MILL</td><td>100A</td><td></td><td>?</td><td>?</td><td>XBI</td><td>◆</td><td>1.</td><td>(19.)</td></tr></table> $\frac{\text{NOM-}\phi \times \text{E10}}{10}$	SNo.	TOOL	NOM- ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	R1	FCE MILL	100A		?	?	XBI	◆	1.	(19.)
SNo.	TOOL	NOM- ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R														
R1	FCE MILL	100A		?	?	XBI	◆	1.	(19.)														
Program type	M																						
Conditions	Immediate																						
Unit	10%																						
Setting range	0 to 9																						

Classification	USER	Display title	LINE/FACE/3D																				
E11	Axial interference clearance of chamfering cutter	The amount of clearance that prevents interference of the chamfering cutter with the bottom during chamfering	 MPL030																				
E12	Radial interference clearance of face milling unit and angular face milling unit	The amount of clearance that prevents interference between the tool and the figure during face milling Example : 	MPL031																				
E13	Tool path setting element for end milling-top unit	Element used to set the tool path internal to the figure for end milling-top unit Example : 	MPL032																				
E14	Depth-of-cut-R automatic setting element (Pocket milling, Pocket milling-mountain, Pocket milling-valley)	Element used to automatically set the radial depth-of-cut (WID-R) of the tool sequence in POCKET, PCKT MT or PCKT VLY unit $\text{WID-R} = \frac{\text{NOM-}\phi \times \text{E14}}{10}$ Example : <table style="margin-left: auto; margin-right: auto;"> <tr> <td>SNo.</td> <td>TOOL</td> <td>NOM-ϕ</td> <td>No.</td> <td>APRCH-X</td> <td>APRCH-Y</td> <td>TYPE</td> <td>ZFD</td> <td>DEP-Z</td> <td>WID-R</td> </tr> <tr> <td>R1</td> <td>END MILL</td> <td>20.</td> <td></td> <td>?</td> <td>?</td> <td>CW</td> <td>G01</td> <td>10.</td> <td>12.</td> </tr> </table> $\frac{\text{NOM-}\phi \times \text{E14}}{10}$	SNo.	TOOL	NOM- ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	R1	END MILL	20.		?	?	CW	G01	10.	12.	↑ ↑
SNo.	TOOL	NOM- ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R														
R1	END MILL	20.		?	?	CW	G01	10.	12.														

Classification	USER	Display title	LINE/FACE/3D								
Address	Meaning	Description									
E15	<p>Tool path setting element for face milling-top unit (reciprocating short)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>10%</td></tr> <tr><td>Setting range</td><td>1 to 9</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	10%	Setting range	1 to 9	<p>Element used to set the tool path external to the defined figure for reciprocating-short machining with face milling unit</p> <p>Example :</p> <p>Tool diameter $\times \frac{E15}{10}$</p> <p>Defined figure MPL033</p>	
Program type	M										
Conditions	Immediate										
Unit	10%										
Setting range	1 to 9										
E16	<p>Peripheral-cutting feed rate override for end milling-mountain unit</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>10%</td></tr> <tr><td>Setting range</td><td>1 to 20</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	10%	Setting range	1 to 20	<p>Override value of the idle-cutting feed rate at which tool of end milling-mountain unit is to be moved around the outer form of the workpiece</p> <p>Note : Valid only when bit 0 of E91 is 1 and its bit 7 is 0.</p> <p>Example :</p> <p>Defined figure</p> <p>FR $\times \frac{E16}{10}$</p> <p>MPL034</p>	
Program type	M										
Conditions	Immediate										
Unit	10%										
Setting range	1 to 20										
E17	<p>Axial-cutting feed rate override</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>10%</td></tr> <tr><td>Setting range</td><td>0 to 9</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	10%	Setting range	0 to 9	<p>Override value of the feed rate at which the tool of a line- or face-machining unit (excluding face milling unit) is to be moved to the machining surface in an axial direction</p> <p>Note :</p> <ol style="list-style-type: none"> Valid only when ZFD (or AFD) of tool sequence is G01. Feed overriding is invalid when this parameter is 0. <p>Example :</p> <p>FR $\times \frac{E17}{10}$</p> <p>SRV-Z</p> <p>Machining surface MPL035</p>	
Program type	M										
Conditions	Immediate										
Unit	10%										
Setting range	0 to 9										
E18	<p>Override in case of the overall width cutting for pocket-machining</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>10%</td></tr> <tr><td>Setting range</td><td>0 to 9</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	10%	Setting range	0 to 9	<p>Override value of feed rate when the pocket-machining radial depth-of-cut becomes equal to the tool diameter</p> <p>Example :</p> <p>FR $\times \frac{E18}{10}$</p> <p>MPL036</p>	
Program type	M										
Conditions	Immediate										
Unit	10%										
Setting range	0 to 9										
		<p>Note :</p> <p>OVERRIDING FOR OVERALL WIDTH CUTTING IS NOT VALID WHEN THIS PARAMETER IS 0.</p> <p>[Applicable units] ROUGH-MACHINING OF POCKET, PCKT MT, PCKT VLY AND STEP</p>									

Classification	USER	Display title	LINE/FACE/3D
Address	Meaning	Description	
E19	Returning feed rate override in case of bidirectional cutting for rough-machining of the end milling-slot unit.	<p>Override value of tool returning feed rate in the rough-machining process of the end milling-slot unit, when the bidirectional cutting is executed.</p> $\text{FR} \times \frac{\text{E19}}{100}$ <p>Note : The override value is invalid, when bit 5 of parameter E96 is set to OFF, or this parameter is set to 0.</p> <p>[Applicable unit] Returning path in rough-machining of SLOT</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 999	
E20	Axial cutting feed override during Z-axial cutting in the pecking mode of face machining	<p>This parameter indicates the feed override value for axial pecking movement to the surface to be machined using a face-machining unit excepting a face milling unit.</p> $\text{Feed rate during pecking operation} = \text{FR} \times \frac{\text{E20}}{10}$ <p>Note :</p> <ol style="list-style-type: none"> Valid only when G01 or a numeric value is set for ZFD (or AFD) and [PECKING] is selected for TYPE in the tool sequence. The feed override function is invalid when "0" is assigned to this parameter. 	
	Program type	M	
	Conditions	Immediate	
	Unit	10%	
	Setting range	0 to 40	
E21	Wall-cutting overlap in closed figure	<p>The amount of overlap of the wall-cutting start and end areas in closed-pattern line- or face-machining</p> <p>Example :</p> <p>MPL037</p> <p>[Applicable units]</p> <ul style="list-style-type: none"> LINE OUT, LINE IN, CHMF OUT and CHMF IN Wall finishing of STEP, POCKET, PCKT MT, PCKT VLY and SLOT 	
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 in	
	Setting range	0 to 999	
E22	Override value of automatic corner over-riding	<p>Override value of automatic corner overriding in line- or face-machining</p> <p>Example :</p> <p>MPL038</p> <p>Note : Automatic corner overriding is invalid when this parameter is 0.</p> <p>[Applicable units]</p> <ul style="list-style-type: none"> LINE RGT, LINE LFT, LINE OUT, LINE IN, STEP, POCKET, PCKT MT and PCKT VLY 	
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 99	

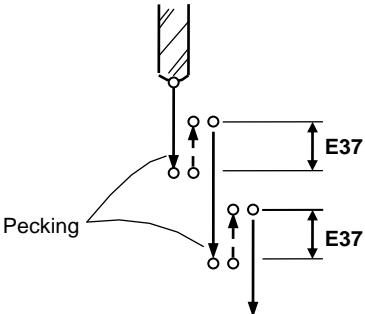
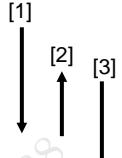
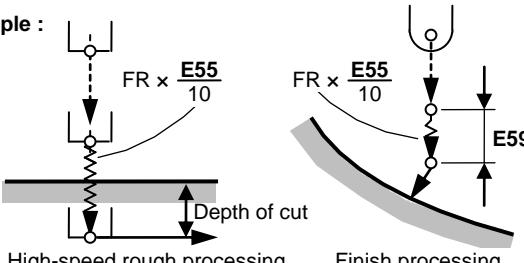
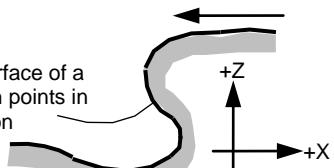
Classification	USER	Display title	LINE/FACE/3D
Address	Meaning	Description	
E23	Effective removal allowance (upper limit) of automatic corner overriding		The range of removal allowances (upper and lower limits) The automatic corner overriding becomes valid when the following line- or face-machining conditions are met: $\text{Tool diameter} \times \frac{\mathbf{E24}}{100} \leq \text{Removal allowance} \leq \text{Tool diameter} \times \frac{\mathbf{E23}}{100}$
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	1 to 99	
E24	Effective removal allowance (lower limit) of automatic corner overriding		 MPL039
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	1 to 99	
E25	Effective angle (upper limit) of automatic corner overriding		The shape angle range (upper limit) The automatic corner overriding becomes valid when the following line- or face-machining conditions are met: $\text{Shape angle} \leq \mathbf{E25}$
	Program type	M	
	Conditions	Immediate	
	Unit	°	
	Setting range	1 to 179	
E26	Calculation coefficient for the finishing feed of line milling		Axial feed rate calculation parameter for a line milling finish. Set the changing ratio of the axial feed rate with respect to the radial feed rate. $\text{Axial feed rate} = \text{Radial feed rate} \times \frac{\mathbf{E26}}{100}$
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 999	

Classification	USER	Display title	LINE/FACE/3D																											
Address	Meaning		Description																											
E27	Radial direction feed rate calculation reference diameter for finish cutting in line machining unit		<p>These parameters, the roughness code, etc. determine the finishing feed rate.</p> <ul style="list-style-type: none"> If the radial-finishing feed rate is taken as F_1, then: $F_1 = \begin{cases} E28 \times \frac{D}{E27 \times \alpha} \times K_f \times Z & (D < E27 \times \alpha) \\ E28 \times K_f \times Z & (D \geq E27 \times \alpha) \end{cases}$ <p>D : Tool diameter α : 0.1 (for metric specs.) or 0.01 (for inch specs.) K_f : Roughness coefficient (Refer to the list below) Z : Number of teeth</p>																											
	Program type	M																												
	Conditions	Immediate																												
	Unit	0.1 mm/0.01 in																												
	Setting range	0 to 65535																												
E28	Finishing feed rate calculation reference feed rate in line machining unit		<table border="1" style="width: 100%; border-collapse: collapse;"> <thead> <tr> <th>Roughness code</th> <th>K_f</th> <th>Roughness code</th> <th>K_f</th> <th>Roughness code</th> <th>K_f</th> </tr> </thead> <tbody> <tr> <td>∇ 1</td> <td>$K_0 \times 0.8^{-3}$</td> <td>$\nabla\nabla$ 4</td> <td>K_0</td> <td>$\nabla\nabla\nabla\nabla$ 7</td> <td>$K_0 \times 0.8^3$</td> </tr> <tr> <td>∇ 2</td> <td>$K_0 \times 0.8^{-2}$</td> <td>$\nabla\nabla\nabla$ 5</td> <td>$K_0 \times 0.8$</td> <td>$\nabla\nabla\nabla\nabla$ 8</td> <td>$K_0 \times 0.8^4$</td> </tr> <tr> <td>$\nabla\nabla$ 3</td> <td>$K_0 \times 0.8^{-1}$</td> <td>$\nabla\nabla\nabla$ 6</td> <td>$K_0 \times 0.8^2$</td> <td>$\nabla\nabla\nabla\nabla$ 9</td> <td>$K_0 \times 0.8^5$</td> </tr> </tbody> </table> <p>K_0=Standard data 0.5</p>				Roughness code	K_f	Roughness code	K_f	Roughness code	K_f	∇ 1	$K_0 \times 0.8^{-3}$	$\nabla\nabla$ 4	K_0	$\nabla\nabla\nabla\nabla$ 7	$K_0 \times 0.8^3$	∇ 2	$K_0 \times 0.8^{-2}$	$\nabla\nabla\nabla$ 5	$K_0 \times 0.8$	$\nabla\nabla\nabla\nabla$ 8	$K_0 \times 0.8^4$	$\nabla\nabla$ 3	$K_0 \times 0.8^{-1}$	$\nabla\nabla\nabla$ 6	$K_0 \times 0.8^2$	$\nabla\nabla\nabla\nabla$ 9	$K_0 \times 0.8^5$
Roughness code	K_f	Roughness code	K_f	Roughness code	K_f																									
∇ 1	$K_0 \times 0.8^{-3}$	$\nabla\nabla$ 4	K_0	$\nabla\nabla\nabla\nabla$ 7	$K_0 \times 0.8^3$																									
∇ 2	$K_0 \times 0.8^{-2}$	$\nabla\nabla\nabla$ 5	$K_0 \times 0.8$	$\nabla\nabla\nabla\nabla$ 8	$K_0 \times 0.8^4$																									
$\nabla\nabla$ 3	$K_0 \times 0.8^{-1}$	$\nabla\nabla\nabla$ 6	$K_0 \times 0.8^2$	$\nabla\nabla\nabla\nabla$ 9	$K_0 \times 0.8^5$																									
Program type	M																													
Conditions	Immediate																													
Unit	0.001 mm/rev 0.0001 in/rev																													
Setting range	0 to 65535																													
E29	Selection of whether the cutting conditions in the shape sequence during VFC mode are to be modified		<p>Select whether the cutting conditions in the shape sequence during VFC mode are to be modified. The selection is valid only for milling.</p> <p>0: Modification of the cutting conditions in the shape sequence 1: No modification of the cutting conditions in the shape sequence</p> <p>In the case of 0, only the cutting conditions in the tool sequence are modified. In the case of 1, both the cutting conditions in the tool sequence and the cutting conditions in the shape sequence are modified.</p>																											
	Program type	M																												
	Conditions	Immediate																												
	Unit	—																												
	Setting range	0, 1																												

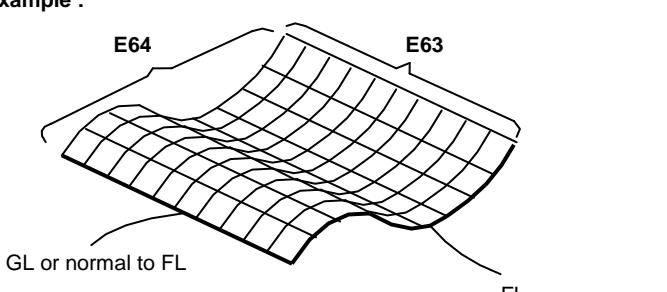
Classification	USER	Display title	LINE/FACE/3D								
E30	<p>Radial clearance for wall attributes during line machining</p> <table border="1" style="margin-left: 10px;"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr><td>Setting range</td><td>0 to 999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.1 mm/0.01 in	Setting range	0 to 999	<p>An element that determines the starting point and escape point of radial cutting when CLOSED is specified for the wall attributes at the starting point and ending point of open-pattern line machining.</p> <p>When OPEN is specified, E2 is used.</p> <p><CLOSED specified></p> <p><OPEN specified></p>	
Program type	M										
Conditions	Immediate										
Unit	0.1 mm/0.01 in										
Setting range	0 to 999										
E31	<p>Element that determines the amount of OPEN attribute wall protrusion for pocket-machining shape units</p> <table border="1" style="margin-left: 10px;"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>10%</td></tr> <tr><td>Setting range</td><td>0 to 9</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	10%	Setting range	0 to 9	<p>Element that determines the amount of OPEN attribute wall protrusion in pocket-machining shape units (POCKET, PCKT MT or PCKT VLY unit only)</p> $\text{CLOSED attribute wall} = \frac{\text{Tool dia. (Note)} \times \text{E31}}{10}$ <p>Note : Nominal tool diameter in the tool sequence is used when tool data is absent.</p>	
Program type	M										
Conditions	Immediate										
Unit	10%										
Setting range	0 to 9										

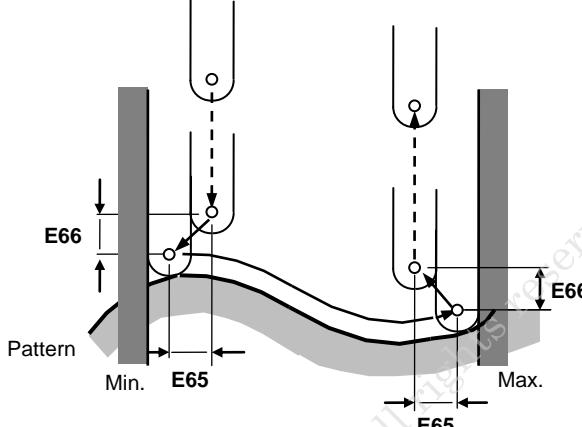
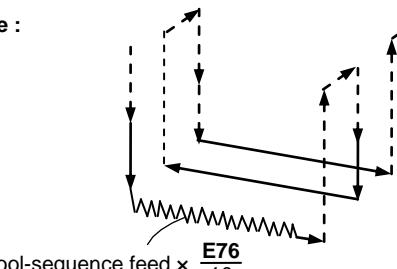
Classification	USER	Display title	LINE/FACE/3D
E32	<p>Element that automatically determines an approaching radius in a Z-direction helical approach scheme.</p> <p>The approaching radius value to be used in the Z-direction helical approach scheme for helical machining is automatically determined by the parameter as follows:</p> $\text{Approaching radius} = \frac{\text{Tool dia.} \times \mathbf{E32}}{100}$ <ul style="list-style-type: none"> For helical approach (Helical circle 2 + 1/4) 	<p>Element that automatically determines an approaching radius in a Z-direction helical approach scheme.</p> <p>The approaching radius value to be used in the Z-direction helical approach scheme for helical machining is automatically determined by the parameter as follows:</p> $\text{Approaching radius} = \frac{\text{Tool dia.} \times \mathbf{E32}}{100}$ <ul style="list-style-type: none"> For helical approach (Helical circle 2 + 1/4) 	<p>Approaching radius $\times (\mathbf{E32}/100)$</p> <p>Approaching radius</p> <p>Clearance Z</p> <p>Clearance Z + Cutting allowance Z</p>
E33	<p>Approaching gradient for a Z-direction helical approach scheme</p> $\frac{\text{Distance in Z-direction}}{\text{Distance in XY-plane}}$ <ul style="list-style-type: none"> For helical approach (Helical circle 2 + 1/4) 	<p>Approaching gradient value for a Z-direction helical approach scheme</p> $\frac{\text{Distance in Z-direction}}{\text{Distance in XY-plane}}$ <ul style="list-style-type: none"> For helical approach (Helical circle 2 + 1/4) 	<p>Approaching radius $\times (\mathbf{E33}/100)$</p> <p>Approaching radius</p> <p>Clearance Z</p> <p>Clearance Z + Cutting allowance Z</p>
	<p>Program type M</p> <p>Conditions After stop of movement</p> <p>Unit %</p> <p>Setting range 1 to 999</p>	<p>Program type M</p> <p>Conditions After stop of movement</p> <p>Unit %</p> <p>Setting range 1 to 999</p>	

Classification	USER	Display title	LINE/FACE/3D
Address		Meaning	
E34		<p>Element that automatically determines an approaching distance in a Z-direction tapered approach scheme</p> <p>Program type M</p> <p>Conditions After stop of movement</p> <p>Unit %</p> <p>Setting range 1 to 999</p>	
		<p>The approaching distance value to be used in a Z-direction tapered approach scheme for tapered machining is automatically determined by the parameter as follows:</p> $\text{Approaching distance} = \frac{\text{Tool dia.} \times \mathbf{E34}}{100}$ <ul style="list-style-type: none"> For tapered approach (3 tapers) 	
		<p>Approaching gradient value for a Z-direction tapered approach scheme</p> $\frac{\text{Distance in Z-direction}}{\text{Distance in XY-plane}}$ <ul style="list-style-type: none"> For tapered approach (3 tapers) 	
		<p>Approaching gradient for a Z-direction tapered approach scheme</p> <p>Program type M</p> <p>Conditions After stop of movement</p> <p>Unit %</p> <p>Setting range 1 to 999</p>	
		<p>The escape distance value to be used in the Z-direction tapered escape scheme after tapered machining is automatically determined by the parameter as follows:</p> $\text{Escape distance} = \frac{\text{Tool dia.} \times \mathbf{E36}}{100}$	
		<p>Element that automatically determines an escape distance in the Z-direction tapered escape scheme</p> <p>Program type M</p> <p>Conditions Immediate</p> <p>Unit %</p> <p>Setting range 1 to 100</p>	

Classification	USER	Display title	LINE/FACE/3D
E37	Amount of return of pecking in the Z-axial pecking mode of face machining	This parameter indicates the amount of return of pecking in the Z-axial pecking mode of face machining.	 MPL058
E38	Returning feed rate of pecking in the Z-axial pecking mode of face machining	Set the returning feed rate of the pecking in the Z-axial pecking mode during the face machining.  Note : This value will be handled as 100 if 0 is set here.	Feed rate (G1) F command (G1) E38 (G1) F command
E55	3-D Axial cutting-feed overriding	Feed overriding for cutting a workpiece in an axial direction using a 3-D unit Example :  Note : Feed overriding is invalid when this parameter is 0.	MPL041
E56	3-D Inversion check of curved-surface pattern	This parameter is used to select whether or not an alarm message is to be displayed if the curved surface of a defined pattern points in the -Z direction (normally, processing becomes impossible). 0: No alarm 1: Alarm Example :  The curved surface of a defined pattern points in the -Z direction	MPL042

Classification	USER	Display title	LINE/FACE/3D
Address		Meaning	
E57		3-D Severity check of cutting pitch	
		Program type	M
		Conditions	Immediate
		Unit	—
		Setting range	0, 1
E58		3-D Tool-diameter compensation	
		Program type	M
		Conditions	Immediate
		Unit	—
		Setting range	0, 1
E59		3-D Allowance of axial-cutting start position	
		Program type	M
		Conditions	Immediate
		Unit	0.1 mm/0.01 in
		Setting range	0 to 999
E60		3-D Normal cutting allowance	
		Program type	M
		Conditions	Immediate
		Unit	0.1 mm/0.01 in
		Setting range	0 to 999
		<p>This parameter is used to select whether or not processing is to be performed in strict accordance with the tool-sequence pitch data setting.</p> <p>0: The pitch setting is not strictly observed. 1: The pitch setting is strictly observed.</p> <p>Note :</p> <ol style="list-style-type: none"> 1. This parameter is invalid during high-speed rough processing. 2. The operation time becomes long if this parameter is set to 1. 	
		<p>This parameter is used to select whether or not 3-D tool-diameter compensation according to tool data is to be made for the curved surface of a defined pattern</p> <p>0: Diameter compensation made 1: Diameter compensation not made</p> <p>Example :</p>	
		<p>MPL043</p>	
		<p>Element used to set the position in which the cutting feed in axial direction is to be started after the tool has been moved from the initial point toward the workpiece at a rapid feed rate</p> <p>Example :</p>	
		<p>MPL044</p>	
		<p>The 3-D finish processing (cutting) allowance in the direction of the normal with respect to the defined pattern of the curved surface</p> <p>Example :</p>	
		<p>MPL045</p>	

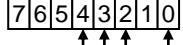
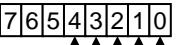
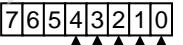
Classification	USER	Display title	LINE/FACE/3D
Address	Meaning	Description	
E61	3-D Search length for parallel cutting	The length of a short line segment which determines the next approximation point for tool-path creation Depending on the tool-sequence selected: E61 is applicable for //1 or //2, or E62 is applicable for _1 or _2 This value will be handled as 0.1 mm (or 0.01 in) if 0 is set here.	
E62	3-D Search length for right-angle cutting		
E63	3-D Pattern display division segment (FL direction)	The number of segments into which the defined pattern of a curved surface is to be divided for display of the curved-surface pattern on the TOOL PATH CHECK display Example :  Note : This parameter is used for display of a curved-surface pattern, and thus the pattern displayed may slightly differ from the actual pattern of the curved surface to be machined.	
E64	3-D Pattern display division segment (GL direction)		

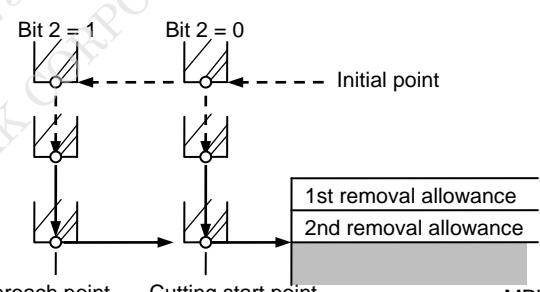
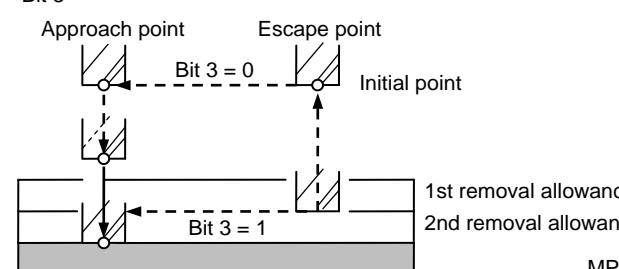
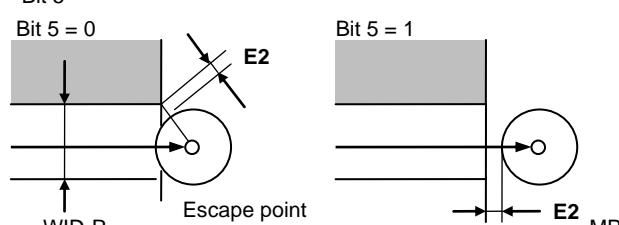
Classification	USER	Display title	LINE/FACE/3D																		
Address		Meaning																			
E65		The allowance of cutting a workpiece along the wall of the area which has been set using the area check function Example :																			
		3-D Radial cutting allowance for area check																			
		Program type	M																		
		Conditions	Immediate																		
		Unit	0.1 mm/0.01 in																		
E66		3-D Axial cutting allowance for area check Example :																			
		3-D Axial cutting allowance for area check																			
		Program type	M																		
		Conditions	Immediate																		
		Unit	0.1 mm/0.01 in																		
E67 to E75		3-D Processing error tolerance Example :																			
		3-D Processing error tolerance	The processing error tolerance with respect to a curved-surface pattern which corresponds to a #T setting (1 through 9) of the tool sequence <table border="1" data-bbox="759 1235 1389 1313"><tr><td>#T</td><td>1</td><td>2</td><td>3</td><td>4</td><td>5</td><td>6</td><td>7</td><td>8</td><td>9</td></tr><tr><td>Address</td><td>E67</td><td>E68</td><td>E69</td><td>E70</td><td>E71</td><td>E72</td><td>E73</td><td>E74</td><td>E75</td></tr></table>	#T	1	2	3	4	5	6	7	8	9	Address	E67	E68	E69	E70	E71	E72	E73
#T	1	2	3	4	5	6	7	8	9												
Address	E67	E68	E69	E70	E71	E72	E73	E74	E75												
Program type	M																				
Conditions	Immediate																				
Unit	0.01 mm/0.001 in																				
E76		The override value which becomes valid in case that the depth-of-cut in a radial direction becomes equal to the entire width (diameter) of the tool Example :																			
		3-D Entire-width override																			
		Program type	M																		
		Conditions	Immediate																		
		Unit	10%																		
		Setting range	0 to 9																		
		Note : Entire-width overriding is not valid when this parameter is 0.																			

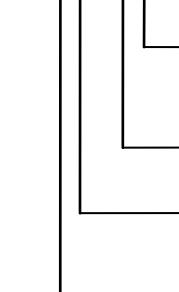
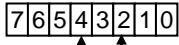
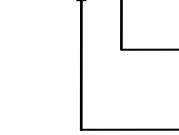
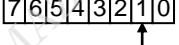
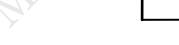
Classification	USER	Display title	LINE/FACE/3D								
Address	Meaning	Description									
E77	<p>3-D Radial cutting allowance for high-speed rough processing (workpiece size appointment)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr><td>Setting range</td><td>0 to 999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.1 mm/0.01 in	Setting range	0 to 999	<p>The clearance of high-speed rough processing (workpiece size appointment) between the tool and the figure</p> <p>Workpiece Defined pattern E77 E77 MPL050</p>	
Program type	M										
Conditions	Immediate										
Unit	0.1 mm/0.01 in										
Setting range	0 to 999										
E78	<p>3-D Multiplying factor set for tolerance</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>%</td></tr> <tr><td>Setting range</td><td>0 to 100</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	%	Setting range	0 to 100	0: 100%	
Program type	M										
Conditions	Immediate										
Unit	%										
Setting range	0 to 100										
E83	<p>3-D Region of radial machining during high-speed rough processing (offset appointment)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr><td>Setting range</td><td>0 to 9999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.1 mm/0.01 in	Setting range	0 to 9999	<p>That amount of offset from a curved-surface pattern which determines the region of high-speed rough processing (offset appointment) in a radial direction</p> <p>Region to be machined Material height Curved-surface pattern E83 MPL051</p>	
Program type	M										
Conditions	Immediate										
Unit	0.1 mm/0.01 in										
Setting range	0 to 9999										
E84	<p>3-D Region of axial machining during high-speed rough processing (offset appointment)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr><td>Setting range</td><td>0 to 9999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.1 mm/0.01 in	Setting range	0 to 9999	<p>That distance from the bottom of a curved-surface pattern which determines the region of high-speed rough processing (offset appointment) in an axial direction</p> <p>Example :</p> <p>Region to be machined Material height Curved-surface pattern E83 E84 MPL052</p>	
Program type	M										
Conditions	Immediate										
Unit	0.1 mm/0.01 in										
Setting range	0 to 9999										

Classification	USER	Display title	LINE/FACE/3D								
Address	Meaning	Description									
E85	<p>3-D Region of radial machining during high-speed rough processing: -X (workpiece size appointment)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr><td>Setting range</td><td>0 to 9999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.1 mm/0.01 in	Setting range	0 to 9999	<p>The factor that determines the region of high-speed rough processing (workpiece size appointment) in a radial direction</p> <p>Curved-surface pattern</p> <p>Region to be machined</p>	
Program type	M										
Conditions	Immediate										
Unit	0.1 mm/0.01 in										
Setting range	0 to 9999										
E86	<p>3-D Region of radial machining during high-speed rough processing: +X (workpiece size appointment)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr><td>Setting range</td><td>0 to 9999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.1 mm/0.01 in	Setting range	0 to 9999		MPL053
Program type	M										
Conditions	Immediate										
Unit	0.1 mm/0.01 in										
Setting range	0 to 9999										
E87	<p>3-D Region of radial machining during high-speed rough processing: -Y (workpiece size appointment)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr><td>Setting range</td><td>0 to 9999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.1 mm/0.01 in	Setting range	0 to 9999		
Program type	M										
Conditions	Immediate										
Unit	0.1 mm/0.01 in										
Setting range	0 to 9999										
E88	<p>3-D Region of radial machining during high-speed rough processing: +Y (workpiece size appointment)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr><td>Setting range</td><td>0 to 9999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.1 mm/0.01 in	Setting range	0 to 9999		
Program type	M										
Conditions	Immediate										
Unit	0.1 mm/0.01 in										
Setting range	0 to 9999										

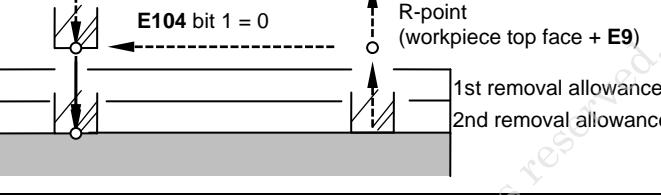
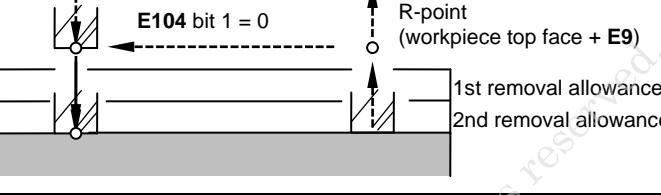
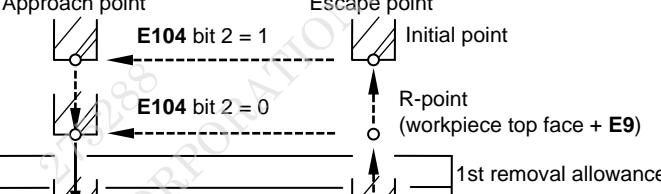
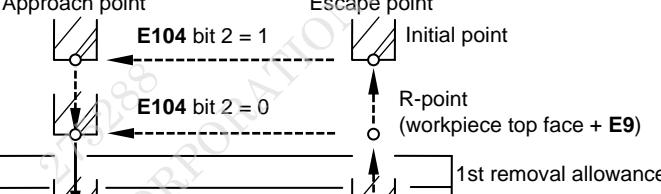
Classification	USER	Display title	LINE/FACE/3D
Address	Meaning	Description	
E89	3-D Region of axial machining during high-speed rough processing (workpiece size appointment)	<p>That distance from the bottom of a curved-surface pattern which determines the region of high-speed rough processing (workpiece size appointment) in an axial direction</p> <p>Example :</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 in	
	Setting range	0 to 9999	MPL054
E91	Tool-path pattern selection for end milling-mountain unit	<p>Bit 7 6 5 4 3 2 1 0</p> <ul style="list-style-type: none"> { 0: Machining from inside to outside 1: Machining from outside to inside { 0: Cutting direction inverted 1: Cutting direction fixed { 0: The R-point height is set always to E9. 1: The R-point height is set to E7 or E9 when there is or isn't pre-machining in the same unit, respectively. { 0: The clearance on X-Y plane is set always to E2. 1: The clearance on X-Y plane is set to E5 or E2 when there is or isn't pre-machining in the same unit, respectively. 1: Rapid feed up to the intended surface + E9 { 0: Tool path based on inside shape 1: Tool path based on outside shape <p>Note :</p> <ol style="list-style-type: none"> 1. If bit 0 = 0, tool path based on inside shape is selected automatically, irrespective of value of bit 7. 2. If bit 0 = 1 and bit 7 = 0, fixed direction of cutting is selected automatically, irrespective of value of bit 1. 3. Bit 4 becomes valid only for two or more rounds of cutting. 	MPL055
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER	Display title	LINE/FACE/3D
Address	Meaning		Description
E92	Tool-path pattern selection for pocket milling unit		<p>{ 0: Machining from inside to outside 1: Machining from outside to inside}</p> <p>{ 0: The R-point height is set always to E9. 1: The R-point height is set to E7 or E9 when there is or isn't pre-machining in the same unit, respectively.</p> <p>{ 0: The clearance on X-Y plane is set always to E2. 1: The clearance on X-Y plane is set to E5 or E2 when there is or isn't pre-machining in the same unit, respectively.</p> <p>1: Rapid feed up to the intended surface + E9</p>
E93	Tool-path pattern selection for pocket milling-mountain unit		<p>{ 0: Machining from inside to outside 1: Machining from outside to inside}</p> <p>{ 0: Cutting direction inverted 1: Cutting direction fixed}</p> <p>{ 0: The R-point height is set always to E9. 1: The R-point height is set to E7 or E9 when there is or isn't pre-machining in the same unit, respectively.</p> <p>{ 0: The clearance on X-Y plane is set always to E2. 1: The clearance on X-Y plane is set to E5 or E2 when there is or isn't pre-machining in the same unit, respectively.</p> <p>1: Rapid feed up to the intended surface + E9</p>
E94	Tool-path pattern selection for pocket milling-valley unit		<p>{ 0: Machining from inside to outside 1: Machining from outside to inside}</p> <p>{ 0: Cutting direction inverted 1: Cutting direction fixed}</p> <p>{ 0: The R-point height is set always to E9. 1: The R-point height is set to E7 or E9 when there is or isn't pre-machining in the same unit, respectively.</p> <p>{ 0: The clearance on X-Y plane is set always to E2. 1: The clearance on X-Y plane is set to E5 or E2 when there is or isn't pre-machining in the same unit, respectively.</p> <p>1: Rapid feed up to the intended surface + E9</p>

Classification	USER	Display title	LINE/FACE/3D
Address	Meaning	Description	
E95	Tool-path pattern selection for line machining unit	<p>Bit 7 Bit 6 Bit 5 Bit 4 Bit 3 Bit 2 Bit 1 Bit 0</p> <ul style="list-style-type: none"> In case of a collision in the line-machining approach path: <ul style="list-style-type: none"> 0: Machining not aborted (conventional method) 1: Machining aborted For the 2nd and subsequent rounds of cutting: <ul style="list-style-type: none"> 0: Not via the approach point 1: Via the approach point For the 2nd and subsequent rounds of cutting: <ul style="list-style-type: none"> 0: Escape to the Z-axis initial point 1: No escape on the Z-axis 1: Rapid feed up to the intended surface + E9 1: Escape is set to a point where the tool comes out of the removal allowance. The R-point height for central, right hand, left hand, outside and inside linear machining is: <ul style="list-style-type: none"> 0: Set always to E9 1: Set to E7 or E9 when there is or isn't pre-machining in the same unit, respectively. The X-Y plane clearance for outside and inside linear machining is: <ul style="list-style-type: none"> 0: Set always to E2 1: Set to E5 or E2 when there is or isn't pre-machining in the same unit, respectively. 	
		<ul style="list-style-type: none"> Bit 2  	MPL501
		<ul style="list-style-type: none"> Bit 3  	MPL502
		<ul style="list-style-type: none"> Bit 5  	MPL503
		<p>Note :</p> <ol style="list-style-type: none"> 1. Bit 3 valid only for inside/outside line machining unit. 2. When bit 3 is set to 1, "no escape on the Z-axis" is selected regardless of bit 2 of parameter E104. 	
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER	Display title	LINE/FACE/3D
Address	Meaning		Description
E96	Tool-path pattern selection for end milling-slot unit Program type M Conditions Immediate Unit Bit Setting range Binary, eight digits	 	<p>0: The R-point height is set always to E9. 1: The R-point height is set to E7 or E9 when there is or isn't pre-machining in the same unit, respectively.</p> <p>For the 2nd and subsequent rounds of cutting: 0: Not via the approach point 1: Via the approach point</p> <p>1: Rapid feed up to the intended surface + E9</p> <p>Returning feed rate override of the end milling-slot unit 0: Invalid 1: Valid</p>
E97	Tool-path pattern selection for end milling-top unit Program type M Conditions Immediate Unit Bit Setting range Binary, eight digits	 	<p>0: The R-point height is set always to E9 1: The R-point height is set to E7 or E9 when there is or isn't pre-machining in the same unit, respectively.</p> <p>1: Rapid feed up to the intended surface + E9</p>
E98	Cutting method selection for end milling-mountain, pocket milling-valley unit Program type M Conditions Immediate Unit Bit Setting range Binary, eight digits	 	<p>1: The 1st cutting amount exceeds the command value at end milling-mountain or pocket valley-machining.</p>
E99 (bit 0)	Milling feed rate specification range for the shape sequence of the MAZATROL program Milling feed rate specification range for the shape sequence of the MAZATROL program Program type M Conditions Immediate Unit Bit Setting range 0, 1		<p>Milling feed rate specification range for the shape sequence of the MAZATROL program 0: Valid for roughing 1: Valid for finishing</p>

Classification	USER	Display title	LINE/FACE/3D
Address	Meaning		Description
E99 (bit 1)	Bottom/wall simultaneous finishing mode for pocket machining		Bottom/wall simultaneous finishing mode for pocket machining 0: Sequential (Conventional scheme) 1: Simultaneous
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
E99 (bit 3)	Interference check between the approach point and the cutting starting point of face machining		Interference check between the approach point and the cutting starting point of face machining 0: Invalid 1: Valid
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
E99 (bit 4)	Alarm generating condition during the approach or escape for line machining		Set whether or not to display alarm 705 APPROACH POINT ERROR and stop operation when the amount of tool movement is 0 for the command in the last line of shape sequence in the right-hand/left-hand linear machining or right-hand/left-hand chamfering unit. 0: Not set 1: Set
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
E104 (bit 0)	Cutting method to be used after an approach point is automatically determined during line/face machining		0: Same operation before and after automatic determination 1: Operation by manual input after automatic determination
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	

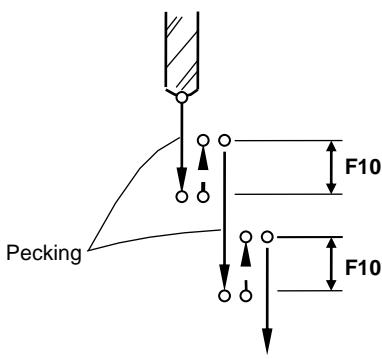
Classification	USER	Display title	LINE/FACE/3D
Address	Meaning	Description	
E104 (bit 1)	Returning position during face machining.	<p>Tool retraction position after each cutting plunge in the Z-axis direction in the face machining unit</p> <p>0: R-point 1: Initial point</p> 	<p>Tool retraction position after each cutting plunge in the Z-axis direction in the face machining unit</p> <p>0: R-point 1: Initial point</p> 
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
	Returning position during line machining.		
E104 (bit 2)	Returning position during line machining.	<p>Tool retraction position after each cutting plunge in the Z-axis direction in the line machining unit</p> <p>0: R-point 1: Initial point</p> 	<p>Tool retraction position after each cutting plunge in the Z-axis direction in the line machining unit</p> <p>0: R-point 1: Initial point</p> 
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
	Selecting an infeed position when a CLOSED attribute is assigned to the wall of line machining		
E104 (bit 3)	0: Starting or ending point of shape + Tool radius + E30	<p>0: Starting or ending point of shape + Tool radius + E30</p> <p>1: Starting or ending point of shape</p>	<p>0: Starting or ending point of shape + Tool radius + E30</p> <p>1: Starting or ending point of shape</p>
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
	Selecting an infeed position when a CLOSED attribute is assigned to the wall of line machining		

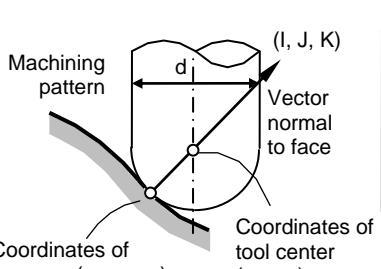
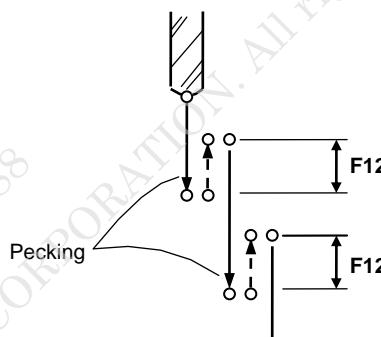
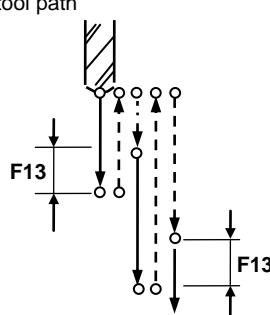
Classification	USER	Display title	LINE/FACE/3D
Address	Meaning		Description
E104 (bit 6)	Calculation of ZC machining feed rate	Program type M Conditions Immediate Unit Bit Setting range 0, 1	0: Calculation based on finishing surface (MATRIX scheme) 1: Calculation based on the starting position of machining (MT Pro scheme)
E105 (bit 0)	Arbitrary B-axis angle during C-axis central line machining	Program type M Conditions Immediate Unit Bit Setting range 0, 1	0: Specifiable 1: Not specifiable
E105 (bit 1)	Machining shape selection method in line machining in ZC mode	Program type M Conditions Immediate Unit Bit Setting range 0, 1	Selection of the surface on which to describe a line machining shape in the ZC mode 0: Developed surface of the cylinder with a radius of SHIFT-R 1: Developed surface of the cylinder with a radius of SHIFT-R + SRV-A
E106 (bit 1)	Selection of a relay point to be used before TPC escapement (line/face machining unit)	Program type M Conditions Immediate Unit Bit Setting range 0, 1	Select the operation to perform before moving to the TPC escape relay point at the end of machining during line/face machining in a program of initial-point scheme. 0: Initial point → TPC escape relay point 1: R-point → TPC escape relay point * Valid only with the machines equipped with ATC-type turret and not equipped with swiveling-type spindle head

2-3-4 User parameter EIA/ISO (F)

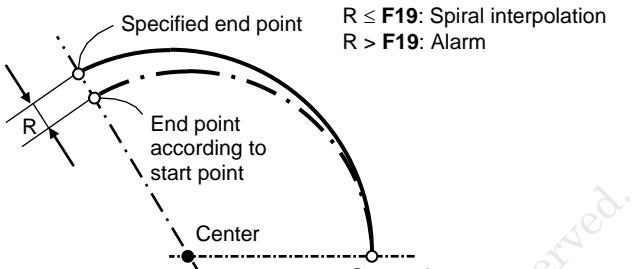
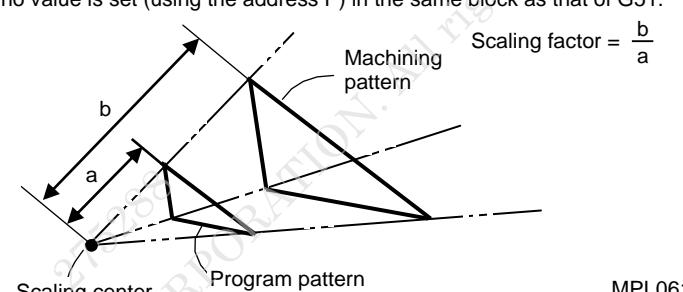
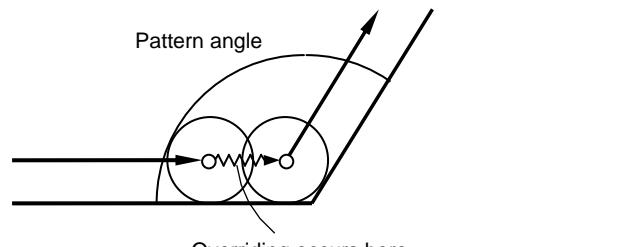
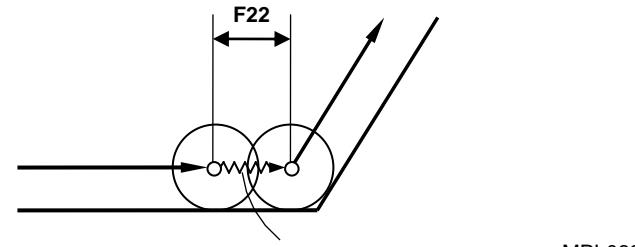
Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F1	G61.1 corner deceleration coefficient (%)		<p>When $(L74/L75) \times (F1/100)$ is assigned as the allowable acceleration attained before corner deceleration is started, the theoretical value V_o of the corner deceleration in G61.1 will be changed to the following V_o' value:</p> $V_o' = V_o \times F1/100$ <p>Note :</p> <p>The assigned value is invalid if minus. If 0 is assigned, the deceleration coefficient will be regarded as 100, or if more than 500 is assigned, the deceleration coefficient will be regarded as 500.</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	
F2	G61.1 arc-clamping speed coefficient (%)		<p>When $(L74/L75) \times (F2/100)$ is assigned as the allowable acceleration attained before arc radius speed clamping is started, the theoretical value V_c of the arc radius clamping speed in G61.1 will be changed to the following V_c' value:</p> $V_c' = V_c \times \sqrt{F2/100}$ <p>Note :</p> <p>The assigned value is invalid if minus. If 0 is assigned, the deceleration coefficient will be regarded as 100, or if more than 500 is assigned, the deceleration coefficient will be regarded as 500.</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	
F3	High-speed smoothing control		<p>0: High-speed smoothing control invalid 1: High-speed smoothing control valid (No deceleration at very slightly stepped sections) 3: High-speed smoothing control invalid (Deceleration at very slightly stepped sections)</p>
	Program type	E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0, 1, 3	

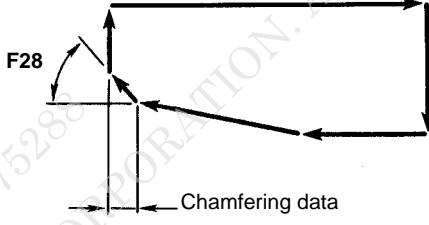
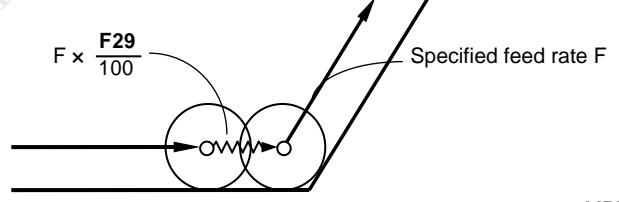
Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F4	Distance to judge the shape formed by 3 axes under tool tip point control in the 5-axis fairing function		Set the distance, succeeding and preceding the point to be modified, to judge the shape formed by the 3 axes under tool tip point control in the 5-axis fairing function Note : When 0 is set, the distance is regarded as 1 mm/0.04 in.
	Example : When the setting is F4 = 5000 [0.5 mm (0.02 in)]		
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 in	
F5	Distance to judge the shape formed by rotary 2 axes in the 5-axis fairing function		Set the distance, succeeding and preceding the point to be modified, to judge the shape formed by the rotary 2 axes in the 5-axis fairing function Note : When 0 is set, the distance is regarded as 1°.
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001°	
	Setting range	0 to 899999	
F6	Minimum allowable height of stepped sections for deceleration in high-speed smoothing control mode		Specify the minimum height of stepped sections at which the axis feed is not to be decelerated in high-speed smoothing control mode.
	Program type	E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 in	
	Setting range	0 to 100	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F7	—		Fixed value (0)
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F8	Corner deceleration speed coefficient for high-speed smoothing control		Corner deceleration speed V_o determined by the optimal corner deceleration function is adjusted using the value set in this parameter. $V_o = V_o \times \frac{F8}{100}$ The speed is not adjusted if 0 is set here. (Setting is handled as 100.) Setting more than 2000 is disposed as 2000.
	Program type	E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 2000	
F9	Circular cutting clamp speed coefficient for high-speed smoothing control		Circular cutting clamp speed V_c is adjusted using the value set in this parameter. $V_c = V_c \times \sqrt{\frac{F9}{100}}$ The speed is not adjusted if 0 is set here. (Setting is handled as 100.) Setting more than 500 is disposed as 500.
	Program type	E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	
F10	Return amount of pecking in the tool path of tap high-speed deep-hole cycle		Return amount during tap high-speed deep-hole cycle
	Return amount during tap high-speed deep-hole cycle		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 in	
	Setting range	0 to 99999999	

Classification	USER	Display title	EIA/ISO								
F11	<p>Vector constant for 3-D tool-diameter compensation</p> <table border="1"> <tr> <td>Program type</td><td>—</td></tr> <tr> <td>Conditions</td><td>Next block</td></tr> <tr> <td>Unit</td><td>0.001 mm/0.0001 in (0.001°)</td></tr> <tr> <td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	—	Conditions	Next block	Unit	0.001 mm/0.0001 in (0.001°)	Setting range	0 to 99999999	 <p>$F11 = \sqrt{I^2 + J^2 + K^2}$ if this parameter is 0.</p>	$\begin{cases} x = x_0 + \frac{I}{F11} \times \frac{d}{2} \\ y = y_0 + \frac{J}{F11} \times \frac{d}{2} \\ z = z_0 + \frac{K}{F11} \times \frac{d}{2} \end{cases}$ MPL057
Program type	—										
Conditions	Next block										
Unit	0.001 mm/0.0001 in (0.001°)										
Setting range	0 to 99999999										
F12	<p>Return amount of pecking in drill high-speed deep-hole cycle or in G73 tool path</p> <table border="1"> <tr> <td>Program type</td><td>M, E</td></tr> <tr> <td>Conditions</td><td>Next block</td></tr> <tr> <td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr> <td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	M, E	Conditions	Next block	Unit	0.001 mm/0.0001 in	Setting range	0 to 99999999		MPL058
Program type	M, E										
Conditions	Next block										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 99999999										
F13	<p>Allowance amount of rapid-feed stop in deep-hole drilling cycle or in G83/G283</p> <table border="1"> <tr> <td>Program type</td><td>M, E</td></tr> <tr> <td>Conditions</td><td>Next block</td></tr> <tr> <td>Unit</td><td>0.0001 mm/0.00001 in</td></tr> <tr> <td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	M, E	Conditions	Next block	Unit	0.0001 mm/0.00001 in	Setting range	0 to 99999999		MPL059
Program type	M, E										
Conditions	Next block										
Unit	0.0001 mm/0.00001 in										
Setting range	0 to 99999999										
F14	<p>Rotation center of coordinates (axis of abscissa)</p> <table border="1"> <tr> <td>Program type</td><td>—</td></tr> <tr> <td>Conditions</td><td>At power on</td></tr> <tr> <td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr> <td>Setting range</td><td>0 to ±99999999</td></tr> </table>	Program type	—	Conditions	At power on	Unit	0.001 mm/0.0001 in	Setting range	0 to ±99999999		
Program type	—										
Conditions	At power on										
Unit	0.001 mm/0.0001 in										
Setting range	0 to ±99999999										

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F15	Rotation center of coordinates (axis of ordinate)	Vector of coordinate rotation (axis of abscissa)	
F16	Program type	—	
	Conditions	At power on	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to ±99999999	
F17	Horizontal length of coordinate rotation	Vector of coordinate rotation (axis of ordinate)	
	Program type	—	
	Conditions	At power on	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to ±99999999	
F18	Vertical length of coordinate rotation	Angle of coordinate rotation	
	Program type	—	
	Conditions	At power on	
	Unit	0.001°	
	Setting range	0 to ±180000	

Classification	USER	Display title	EIA/ISO
F19	Maximum permissible difference in arc radius	<p>Maximum radius difference that causes spiral interpolation to be performed when the arc-drawing start point and end point radii that have been specified in the arc command do not agree.</p>  <p>R ≤ F19: Spiral interpolation R > F19: Alarm</p>	MPL060
F20	Fixed value of scaling factor	<p>That fixed value of the scaling factor which becomes valid in the case that no value is set (using the address P) in the same block as that of G51.</p>  <p>Scaling factor = $\frac{b}{a}$</p>	MPL061
F21	Maximum inside-corner angle available with automatic corner override (G62)	<p>The automatic corner override using the G62 code becomes valid when the following condition of the shape angle is met:</p> <p>Shape angle ≤ F21</p>  <p>Pattern angle</p> <p>Overriding occurs here.</p>	MPL062
F22	Deceleration area of automatic corner overriding (G62)	<p>The area in which automatic corner overriding using the G62 code occurs</p>  <p>F22</p> <p>Overriding occurs here.</p>	MPL063

Classification	USER	Display title	EIA/ISO		
Address	Meaning	Description			
F27		Handling of G92 (spindle speed clamp value) command at restart Program type E Conditions Immediate Unit — Setting range 0 to 9			
F28		Threading chamfering angle Program type E Conditions Immediate Unit ° Setting range 45, 60			
F29		Override value of automatic corner overriding (G62) Program type E Conditions Next command Unit % Setting range 0 to 100			
F30		G-code type selection Program type E Conditions At power on Unit Decimal number Setting range 0, 1			
0: Only the last G92SsQqRr command is enabled. 1: All the G92SsQqRr commands are enabled. 2 to 8: Not used. 9: The spindle speed clamp value is invalid at restart.					
 F28 Chamfering data					
Note : Set 45 or 60.					
 F29 Specified feed rate F $F \times \frac{F29}{100}$ MPL064					
Note : The automatic corner overriding is invalid when this parameter is 0.					
Select a G-code type. 0: G-code series for machining centers 1: G-code series for turning machines Settings other than the above are all handled as 0.					
Note : Do not disturb the default setting. It is a fixed value for your machine model.					

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F31	Select a roughing priority type for program layout	<p>Select a roughing priority type for program layout (Rearrangement of milling tool sequence).</p> <p>0: Rearranging a roughing tool, a normal tool, and a finishing tool, in that order</p> <p>1: Rearranging a roughing tool, a normal tool, and a finishing tool, in that order Further rearranging each tool with priority given to the same tool (with roughing/finishing tool distinction)</p> <p>2: Milling to be excluded from tool rearrangement.</p> <p>3: Rearranging each tool with priority given to the same tool (with roughing/finishing tool distinction)</p> <p>4: Rearranging a roughing tool, a normal tool, and a finishing tool, in that order Further rearranging normal tools in the order of spot drilling → drilling → end milling → chamfering → boring → reaming → back spot facing → back boring → tapping</p> <p>5: Rearranging a roughing tool, a normal tool, and a finishing tool, in that order Further rearranging normal tools in the order of spot drilling → drilling → end milling → chamfering → boring → reaming → back spot facing → back boring → tapping</p> <p>In addition, for the manual programming unit, assigning priority numbers, judged from the tools to be used</p>	
	Program type	M	
	Conditions	After stop of movement	
	Unit	Decimal number	
	Setting range	0 to 5	
F32	Operation when argument R is omitted from the maximum/minimum spindle speed setting	<p>Select operation when argument R is omitted from the maximum/minimum spindle speed setting.</p> <p>0: Alarm</p> <p>1: Handled as R1</p>	
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F33 (bit 0)	Incorporation of the MAZATROL tool length into the current position counter	<p>Select whether to make the incorporation of the tool length into current-position counter Z valid when tool length correction is in a canceled status, irrespective of whether the program selected on the POSITION display is EIA/ISO or MAZATROL.</p> <p>0: Valid</p> <p>1: Invalid</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F33 (bit 1)	Type of turning tool to be automatically set in the MAZATROL program created by loading an M640T program	Type of the tool to be automatically set in the MATRIX program, created by loading an M640T or T-NEXUS program, for a unit which corresponds to the original units BAR FCE, CPY FCE, EDG FCE, EDG BAK, BAR BAK, and CPY BAK. 0: Facing tool 1: Outside turning tool	
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F33 (bit 4)	Selection of restart setting method in the MAZATROL program	Method of selecting the restart position in the MAZATROL program 0: Selection from program layout 1: By setting [WNo./UNo./SNo REPEAT]	
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F33 (bit 6)	Modal indication of modal group 1 in the fixed cycle mode	0: The indication at the selection of the fixed cycle remains as it is throughout the cycle. (No updating) 1: The indication changes to show the actual modal condition of the cycle operation.	
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F33 (bit 7)	Modal indication on "radius data input for "X-axis" in the mode of 3D coordinate conversion (G68.5)	0: The indication at the G68.5 mode selection remains as it is. 1: G122.1 (radius data input for X-axis ON) is indicated during the G68.5 mode.	
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	

Classification	USER		Display title	EIA/ISO
Address	Meaning		Description	
F34 (bit 0)	Cancellation of the ram spindle offset command (G52.1) when the RESET key is pressed, valid/invalid		0: Invalid (G52.1 hold) 1: Valid	
	Program type	M, E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
F34 (bit 1)	Representation on the POSITION display by the G49 command		The data number and value for tool length offset on the POSITION display upon execution of a G49 (Tool length offset OFF) command 0: Not changed at all 1: Reset to zero (0)	
	Program type	M, E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
F34 (bit 2)	Parts count override in the course of automatic operation		To manually override the parts count (current number of machined workpieces) in the middle of automatic operation 0: Impossible (The operation caused by the ATC and RETURN settings in the end unit occurs with respect to the parts count.) 1: Possible (The operation caused by the ATC and RETURN settings in the end unit occurs irrespectively of the parts count.)	
	Program type	M, E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
F34 (bit 4)	Programming type for inclined-surface machining		0: Type A (5-axis machining available) 1: Type B (MPro compatible)	
	Program type	E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		

Classification	USER		Display title	EIA/ISO
Address	Meaning		Description	
F34 (bit 6)	Compensating movement for a T-code with a movement command in one block		0: No compensating movement on an unspecified axis till completion of the specified movement 1: Simultaneous compensating movement on all the axes including unspecified one, on condition that the tool number is not changed * Bit 6 of F34 is valid only when G-code series T (Turning machines) is selected and F161 bit 2 is set to 1.	
	Program type	M, E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
F34 (bit 7)	Application of Z-OFFSET and C-OFFSET values for the tape operation mode		0: Application of the Z-OFFSET and C-OFFSET values prepared for the program that is currently selected from the STANDARD PROGRAM storage area for the automatic operation 1: No application of the Z-OFFSET and C-OFFSET values prepared on the SETUP INFORMATION display	
	Program type	M, E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
F35 (bit 0)	Multiple material pieces arrangement on the VIRTUAL MACHINING or 3D MONITOR display		Select whether or not multiple material pieces can be arranged on the VIRTUAL MACHINING or 3D MONITOR display. 0: Arrangement of a single piece 1: Arrangement of multiple pieces	
	Program type	M, E		
	Conditions	At power on		
	Unit	Bit		
	Setting range	0, 1		
F35 (bit 1)	Automatic setting of the material model for a MAZATROL program of initial-point scheme on the VIRTUAL MACHINING or 3D MONITOR display		Automatic setting of the material model for a MAZATROL program of initial-point scheme on the VIRTUAL MACHINING or 3D MONITOR display. 0: Valid 1: Invalid	
	Program type	M		
	Conditions	At power on		
	Unit	Bit		
	Setting range	0, 1		

Classification	USER		Display title	EIA/ISO		
Address	Meaning		Description			
F35 (bit 2)	Machine status when assigning G-code for a cutting function not to be represented with the INTELLIGENT SAFETY SHIELD being selected for automatic operation		Machine status when assigning G-code for a cutting function not to be represented with the INTELLIGENT SAFETY SHIELD being selected for automatic operation. 0: Continuing the operation by canceling the interference check function for material model 1: Alarm stop			
	Program type	M, E				
	Conditions	At power on				
	Unit	Bit				
	Setting range	0, 1				
F35 (bit 3)	Machine status when a program prepared and stored for the Hard Disk operation is called up as a subprogram with the INTELLIGENT SAFETY SHIELD being selected for automatic operation		Machine status when a program prepared and stored for the Hard Disk operation is called up as a subprogram with the INTELLIGENT SAFETY SHIELD being selected for automatic operation. 0: Continuing the operation by canceling the INTELLIGENT SAFETY SHIELD for automatic operation 1: Alarm stop			
	Program type	M, E				
	Conditions	At power on				
	Unit	Bit				
	Setting range	0, 1				
F35 (bit 4)	Taking-off a material model's part in the flow of turning operation on the VIRTUAL MACHINING and 3D MONITOR displays		Representation of cutting-off or parting operation by turning on the VIRTUAL MACHINING and 3D MONITOR displays 0: No representation (Cut-off part remains shown in its original place and undergoes the interference check.) 1: Representation (Cut-off part is hidden and excluded from the interference check.)			
	Program type	M, E				
	Conditions	Immediate				
	Unit	Bit				
	Setting range	0, 1				
F35 (bit 6)	Taking-off a material model's part in the flow of milling operation on the VIRTUAL MACHINING and 3D MONITOR displays		Representation of cutting-off or parting operation by milling on the VIRTUAL MACHINING and 3D MONITOR displays 0: No representation (Cut-off part remains shown in its original place and undergoes the interference check.) 1: Representation (Cut-off part is hidden and excluded from the interference check.)			
	Program type	M, E				
	Conditions	Immediate				
	Unit	Bit				
	Setting range	0, 1				

Classification	USER		Display title	EIA/ISO		
Address	Meaning		Description			
F35 (bit 7)	Alarming for a change of feed-overriding value with the INTELLIGENT SAFETY SHIELD being active for automatic operation		Select whether or not it is to be indicated by an alarm message that the feed-overriding value has been changed in the flow of automatic operation with the INTELLIGENT SAFETY SHIELD being active. 0: Alarming 1: No alarming			
	Program type	M, E				
	Conditions	Immediate				
	Unit	Bit				
	Setting range	0, 1				
F36 (bit 1)	Automatic M-code setting in EIA/ISO program modal restarting mode		M-code displayed as search results in EIA/ISO program modal restarting mode 0: Do not set automatically 1: Set automatically (up to six M-codes preceding immediately)			
	Program type	E				
	Conditions	Immediate				
	Unit	Bit				
	Setting range	0, 1				
F36 (bit 2)	Automatic S-code setting in EIA/ISO program modal restarting mode		S-code displayed as search results in EIA/ISO program modal restarting mode 0: Do not set automatically 1: Set automatically (immediately preceding S-code)			
	Program type	E				
	Conditions	Immediate				
	Unit	Bit				
	Setting range	0, 1				
F36 (bit 3)	Error check function in cases where numerical data is omitted in the EIA program or in MDI mode		0: Invalid 1: Valid Example : If numerical data is omitted, such as "G90G0X", alarm 807 ILLEGAL FORMAT is displayed.			
	Program type	E				
	Conditions	Immediate				
	Unit	Bit				
	Setting range	0, 1				

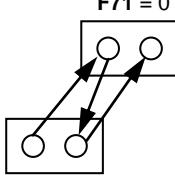
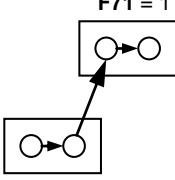
Classification	USER	Display title	EIA/ISO																										
Address	Meaning	Description																											
F36 (bit 4)	Display of the chuck and tailstock areas for an EIA/ISO program on the TRACE or TOOL PATH CHECK display	Whether or not the chuck and tailstock areas are to be displayed for an EIA/ISO program on the TRACE or TOOL PATH CHECK display																											
	Program type	E	0: Not displayed																										
	Conditions	Immediate	1: Displayed																										
	Unit	Bit																											
	Setting range	0, 1																											
F36 (bit 6)	Number of effective decimal digits in the tool axis vector's components (I, J, K) for tool tip point control type 2 (G43.5)	<table border="1"> <thead> <tr> <th>F36 bit 6</th><th>System of units</th><th>Command unit</th><th>Minimum value</th><th>Maximum value</th></tr> </thead> <tbody> <tr> <td>0</td><td>mm</td><td>0.0001</td><td>-99999.9999</td><td>99999.9999</td></tr> <tr> <td></td><td>in</td><td>0.00001</td><td>-9999.99999</td><td>9999.99999</td></tr> <tr> <td>1</td><td>mm</td><td>0.0000001</td><td>-99.9999999</td><td>99.9999999</td></tr> <tr> <td></td><td>in</td><td>0.0000001</td><td>-99.9999999</td><td>99.9999999</td></tr> </tbody> </table>			F36 bit 6	System of units	Command unit	Minimum value	Maximum value	0	mm	0.0001	-99999.9999	99999.9999		in	0.00001	-9999.99999	9999.99999	1	mm	0.0000001	-99.9999999	99.9999999		in	0.0000001	-99.9999999	99.9999999
F36 bit 6	System of units	Command unit	Minimum value	Maximum value																									
0	mm	0.0001	-99999.9999	99999.9999																									
	in	0.00001	-9999.99999	9999.99999																									
1	mm	0.0000001	-99.9999999	99.9999999																									
	in	0.0000001	-99.9999999	99.9999999																									
	Program type	E																											
	Conditions	Immediate																											
	Unit	Bit																											
	Setting range	0, 1																											
F36 (bit 7)	Rotational axis prefiltering	0: Invalid 1: Valid																											
	Program type	E																											
	Conditions	Immediate																											
	Unit	Bit																											
	Setting range	0, 1																											
F37 (bit 0)	Temporary cancellation of geometry compensation for G0 in the mode of tool tip point control	0: Invalid 1: Valid																											
	Program type	E																											
	Conditions	Immediate																											
	Unit	Bit																											
	Setting range	0, 1																											

Classification	USER		Display title	EIA/ISO
Address	Meaning		Description	
F37 (bit 1)	Subprogram call given in manual program unit (for turning machines only)		For turning machines only: Alarming for a subprogram call command given in a manual program (MANL PRG) unit 0: No alarming 1: Alarming	
	Program type	M, E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
F37 (bit 2)	Tool length compensation in MDI operation		Z-axis movement immediately after ATC during MDI operation when the MAZATROL tool length is valid (bit 3 of parameter F93 is set to 1) 0: Perform tool length compensation. 1: Do not perform tool length compensation.	
	Program type	E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
F37 (bit 3)	Timing of clearing #3006 (stop) message		Select the timing of clearing the message that is displayed at #3006 (stop). 0: At the end of automatic operation 1: At the execution of the next block	
	Program type	E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
F37 (bit 6)	Operation when "ATC" is not specified in end unit		Set the operation pattern when "ATC" is not set in end unit 0: Same operation as when "1" is set for "ATC" (The tool is returned and the axes move to the returning position.) 1: Same operation as when "0" is set for "ATC" (The tool is not returned.)	
	Program type	M		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		

Classification	USER		Display title	EIA/ISO		
Address	Meaning		Description			
F38 (bit 1)	Handling of tool index numbers (after tool data exchange)		Tool index number and that of the tool to change when the [TOOLDATA MOVE] menu is selected on the TOOL DATA display for machining centers (without multiple tool data registration) 0: Exchange 1: Do not exchange			
	Program type	M, E				
	Conditions	Immediate				
	Unit	Bit				
	Setting range	0, 1				
F38 (bit 2)	Font size setting for EIA/ISO program display and EIA MONITOR display		0: Large 1: Small			
	Program type	E				
	Conditions	At power back on				
	Unit	Bit				
	Setting range	0, 1				
F38 (bit 4)	Negative tool diameter setting		Negative tool diameter setting in tool data 0: Impossible If a negative value is input, alarm 402 ILLEGAL NUMBER INPUT occurs. 1: Possible * When this parameter is set to "1", alarm 654 TOOL DATA ERROR will occur if a tool to which a negative tool diameter is input is used in the MAZATROL program.			
	Program type	E				
	Conditions	Immediate				
	Unit	—				
	Setting range	0, 1				
F39	Plane to be used for an EIA/ISO program (called by a subprogram unit) and a manual program unit		Default setting of the modal plane selection G-code to be used for an EIA/ISO program (called by a subprogram unit) and a manual program unit. 0: G17 1: G18 2: G19 Settings other than the above are all handled as 0.			
	Program type	M				
	Conditions	Immediate				
	Unit	—				
	Setting range	0 to 2				

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F40	Operating method selection in tape mode		0: Tape operation 1: Hard disc operation 2: IC memory card operation 3: Ethernet operation
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 3	
F41	Threading termination waiting time processing		<p>Set a threading termination waiting time.</p> <p>F41 = 0 No waiting time</p> <p>F41 = 1 to 255 Setting $\times 1.7$ ms</p> <p>Note : An excessively high setting may cause the thread of the run-out section to be destroyed.</p>
	Program type	M	
	Conditions	Immediate	
	Unit	1.7 ms	
	Setting range	0 to 255	
F42	Deceleration area r during Z-axis measurement		<p>Distance (r) between the starting point of movement at measuring speed and the measuring point This data is used when argument R is omitted in G37 command format.</p> <p>G37 Z_ R_r D_ F_ ; (G37)</p>
	Program type	E	
	Conditions	After stop of movement	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	

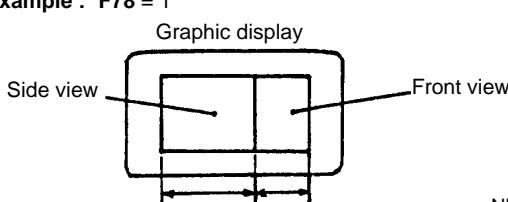
Classification	USER		Display title	EIA/ISO
Address	Meaning		Description	
F43	Measurement area d during Z-axis measurement		Range (d) where the tool should stop This data is used when argument D is omitted in G37 command format. $G37 \text{ } Z_ R_ D_d \text{ } F_;$ (G37)	
	Program type	E		
	Conditions	After stop of movement		
	Unit	0.001 mm/0.0001 in		
	Setting range	0 to 99999999		
F44	Measuring speed f		Measuring speed (f) This data is used when argument F is omitted in G37 command format. $G37 \text{ } Z_ R_ D_ F_f;$ (G37)	
	Program type	E		
	Conditions	After stop of movement		
	Unit	mm/min/0.1 in/min		
	Setting range	0 to 120000		
F45	Deceleration area r during X-axis measurement		Distance (r) between the starting point of movement at measuring speed and the measuring point This data is used when argument R is omitted in G37 command format. $G37 \text{ } X_ R_r \text{ } D_ F_;$ (G37)	
	Program type	E		
	Conditions	After stop of movement		
	Unit	0.001 mm/0.0001 in		
	Setting range	0 to 99999999		
F46	Deceleration area d during X-axis measurement		Range (d) where the tool should stop This data is used when argument D is omitted in G37 command format. $G37 \text{ } X_ R_ D_d \text{ } F_;$ (G37)	
	Program type	E		
	Conditions	After stop of movement		
	Unit	0.001 mm/0.0001 in		
	Setting range	0 to 99999999		

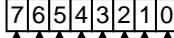
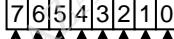
Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F47 to F66	Common variable name Program type — Conditions — Unit — Setting range —	The name of the variable specified by the user macro SETVN is displayed (Name display only. No setting is possible on the PARAMETER display.) F47: Name of #500 ⋮ ⋮ F66: Name of #519	
F69	EIA/ISO program restart method Program type E Conditions Immediate Unit — Setting range 0, 1	This parameter is used to select the method of specifying the EIA/ISO program restarting position. Two methods are available: 0: The whole program, including the subprograms, is subjected to this processing. Set the sequence number, block number and number of times of repetition as searched from the beginning part of the main program. 1: The subprogram including the desired restart position can be specified. After setting the work number of the corresponding program, set the sequence number, block number, and number of times of repetition as searched from the beginning part.	
F70	Availability of multiple-machining and designated number of repetitions in the EIA/ISO subprogram Program type M Conditions Immediate Unit — Setting range 0, 1	When the EIA/ISO program is called up as a subprogram, this parameter is used to validate/invalidate multiple-machining and the specified number of times to restart the program. 0: Multiple-machining is ineffective for the EIA/ISO subprogram. Specified number of times to restart the EIA/ISO subprogram is effective. 1: Multiple-machining is effective for the EIA/ISO subprogram. Specified number of times to restart the EIA/ISO subprogram is ineffective.	
F71	Machining order control Program type M Conditions Immediate Unit — Setting range 0, 1	Tool priority and multiple-machining priority selection 0: Identical-tool priority function is executed first. 1: Multiple-machining function is executed first. Example : Multiple-machining of two workpieces using a spot drill	<p style="text-align: center;">F71 = 0</p>  <p style="text-align: center;">F71 = 1</p>  <p style="text-align: right;">MPL065</p>

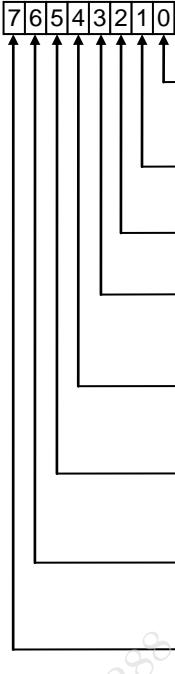
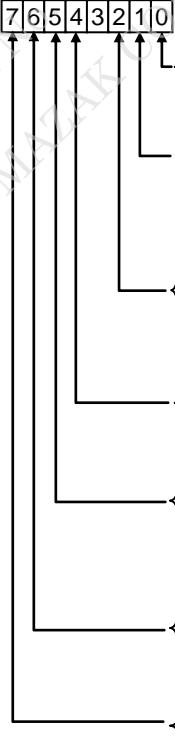
Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F72	Selection of the shape correction function of the MAZATROL program Program type — Conditions — Unit — Setting range 0, 1	To select whether the shape correction function of the MAZATROL program is always effective or ineffective. 0: Invalid 1: Shape correction function valid	
F73	M-code execution time for time study Program type M, E Conditions Immediate Unit 0.01 s Setting range 0 to 10000	The tool-path check time study time that is accumulated each time an M-code is output.	
F74	S-code execution time for time study Program type M, E Conditions Immediate Unit 0.01 s Setting range 0 to 10000	The tool-path check time study time that is accumulated each time an S-code is output.	
F75	T-code execution time for time study Program type M, E Conditions Immediate Unit 0.01 s Setting range 0 to 10000	The tool-path check time study time that is accumulated each time a T-code is output.	

2

PARAMETER

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F76	B-code execution time for time study		The tool-path check time study time that is accumulated each time a B-code is output.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.01 s	
	Setting range	0 to 10000	
F77	Basis rate for tool life judgment		<p>The basis rate for the NC to judge whether the tool is to be displayed in reverse display mode on the TOOL DATA display to indicate that the life of that tool is approaching expiry.</p> <p>If bit 2 in F82 is 0: When the rate of the operation time to the estimated life exceeds the setting of the F77 parameter, the NC will judge the tool to be approaching expiry.</p> <p>If bit 2 in F82 is 1: When the residual life decreases below the setting of the F77 parameter, the NC will judge the tool to be approaching expiry.</p> <p>Note : The above judgment function is invalid if this parameter is set to 0.</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	%, min	
	Setting range	0 to 9999	
F78	Selection of separating ratio of graphic display		<p>Selection of separation ratio between side view and front view (or rear view) when two split plane indication mode has been selected.</p> <p>F78 = 0 1:1 F78 = 1 2:1 F78 = 2 5:1</p> <p>Example : F78 = 1</p>  <p>NM211-00217</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1, 2	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F79	—		 <ul style="list-style-type: none"> Holding of memory monitor address 0: No 1: Yes Selection of menu display <ul style="list-style-type: none"> 0: Menu for machining centers 1: Menu for turning centers Key history function <ul style="list-style-type: none"> 0: Yes 1: No Tool search method <ul style="list-style-type: none"> 0: In order of TNo. 1: In order of TNo. of tools currently in use <ul style="list-style-type: none"> * Only bit 3 becomes valid after power-on. Selection of tap gear <ul style="list-style-type: none"> 0: M32 system 1: M640M Pro system (M640M/M PLUS system) Display of tools currently in use <ul style="list-style-type: none"> 0: No 1: Yes Initial value of synchronous/asynchronous tapping during tapping tool registration <ul style="list-style-type: none"> 0: Synchronous tapping 1: Asynchronous tapping <ul style="list-style-type: none"> * This parameter is valid only when a sync tapping option is provided. Display of a MAZATROL monitor window <ul style="list-style-type: none"> 0: Yes 1: No
	Program type	M, E	
	Conditions	Immediate (Bit 3: At power on)	
	Unit	Bit	
	Setting range	Binary, eight digits	
F80	—		 <ul style="list-style-type: none"> MAZATROL function <ul style="list-style-type: none"> 0: Valid 1: Invalid (Only EIA-related displays valid) Automatic display of the navigation window on the occurrence of an alarm <ul style="list-style-type: none"> 0: Display off 1: Display on MAINTENANCE CHECK display at power on <ul style="list-style-type: none"> 0: Not displayed 1: Displayed Third page of the MAINTENANCE CHECK display <ul style="list-style-type: none"> 0: Not displayed 1: Displayed Automatic display of the GRAPHIC MAINTENANCE display on the occurrence of an alarm <ul style="list-style-type: none"> 0: Display off 1: Display on Learning of cutting conditions <ul style="list-style-type: none"> 0: Invalid 1: Valid Editing on the CUTTING CONDITION LEARN display <ul style="list-style-type: none"> 0: Invalid 1: Valid Destination of spare tool correction by the workpiece measurement <ul style="list-style-type: none"> 0: Tool length and tool diameter of the TOOL DATA display 1: Tool length and tool diameter correction of the TOOL DATA display
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

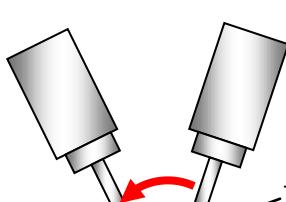
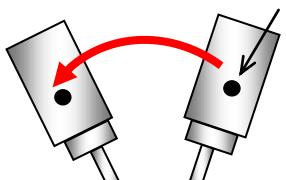
Classification	USER	Display title	EIA/ISO								
Address	Meaning	Description									
F81	<p>—</p> <table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>At power on</td></tr> <tr><td>Unit</td><td>Bit</td></tr> <tr><td>Setting range</td><td>Binary, eight digits</td></tr> </table>	Program type	M, E	Conditions	At power on	Unit	Bit	Setting range	Binary, eight digits	 <ul style="list-style-type: none"> Program management function <ul style="list-style-type: none"> 0: Normal 1: Editing prohibited (in 9000's) Program management function <ul style="list-style-type: none"> 0: Normal 1: Displaying prohibited (in 9000's) Fixed value (0) Add-in MAZATROL <ul style="list-style-type: none"> 0: Invalid 1: Valid Program management function <ul style="list-style-type: none"> 0: Normal 1: Editing prohibited (in 8000, 9000's) Program management function <ul style="list-style-type: none"> 0: Normal 1: Displaying prohibited (in 8000, 9000's) Retaining the program transfer settings <ul style="list-style-type: none"> 0: Invalid 1: Valid Input error prevention function <ul style="list-style-type: none"> 0: Invalid 1: Valid 	
Program type	M, E										
Conditions	At power on										
Unit	Bit										
Setting range	Binary, eight digits										
F82	<p>—</p> <table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Bit</td></tr> <tr><td>Setting range</td><td>Binary, eight digits</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Bit	Setting range	Binary, eight digits	 <ul style="list-style-type: none"> 0: Characteristics estimation result graph display off 1: Characteristics estimation results graph display on Selection of inch/metric representation in POSITION display, TOOL DATA display, and TOOL OFFSET display modes <ul style="list-style-type: none"> 0: Invalid 1: Valid Basis for tool life judgment <ul style="list-style-type: none"> 0: Whether the rate of the operation time to the estimated life is greater than the setting of the F77 parameter 1: Whether the residual life is less than the setting of the F77 parameter X-axis diameter display on the POSITION display <ul style="list-style-type: none"> 0: Invalid 1: Valid Specify whether the stored tools registration function on the VISUAL TOOL MANAGEMENT display is to be made valid or invalid when the visual tool ID/data management functions are valid <ul style="list-style-type: none"> 0: Invalid 1: Valid Select whether to make the incorporation of tool nose width valid when grooving tool length is auto-set for the B-axis type of turret. <ul style="list-style-type: none"> 0: Invalid 1: Valid Index tool wear correction setup during automatic operation <ul style="list-style-type: none"> 0: Impossible 1: Possible <p>Note :</p> <ol style="list-style-type: none"> Bit 1 in parameter F82 is valid only when an inch/metric unit system change simplifying function (option) is provided. Bit 4 in parameter F82 is valid only for the X-axis for which bit 5 in M18 is set to 0, and the diameter value is displayed in turning mode. (Only for the INTEGREX e Series) 	
Program type	M										
Conditions	Immediate										
Unit	Bit										
Setting range	Binary, eight digits										

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F83	—		<p>Output of the alarm history data as text data 0: Valid 1: Invalid</p> <p>Operation record function 0: Invalid 1: Valid</p> <p>Select the attribute of the starting/ending point of line machining after loading a program of M640 series onto the MATRIX (only for a program of Z-offset scheme) 0: CLOSED 1: OPEN</p>
F84 (bit 0)	Selection of whether or not the tool offset data is taken into account for the POSITION counter during the execution of EIA/ISO programs	<p>Program type M, E</p> <p>Conditions Immediate</p> <p>Unit Bit</p> <p>Setting range Binary, eight digits</p>	<p>Select whether or not the tool offset data is taken into account for the POSITION counter during automatic operation or virtual machining with EIA/ISO programs</p> <p>0: Not taken into account 1: Taken into account</p> <p>* In the tool path check, the tool offset value is not taken into account disregarding the setting of this parameter.</p>
F84 (bit 1)	Switching of arguments B and J in a fixed cycle	<p>Program type E</p> <p>Conditions At power on</p> <p>Unit —</p> <p>Setting range 0, 1</p>	<p>Switches the argument to be used in a fixed cycle program.</p> <p>0: B is used. 1: J is used.</p>
F84 (bit 2)	Spare tool search method for EIA/ISO programs	<p>Program type E</p> <p>Conditions At power on</p> <p>Unit —</p> <p>Setting range 0, 1</p>	<p>0: Group number assignment 1: Tool number assignment</p>

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F84 (bit 3)	Timing to validate new workpiece offset data specified with a system variable		0: Valid when the workpiece offset is specified after a system variable is entered. 1: Valid immediately after a system variable is entered.
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
F84 (bit 4)	Machine coordinate system (G92) selection		0: M32, M PLUS, M640M, or M640M Pro system 1: M2 system
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
F84 (bit 5)	Incremental/absolute data command in high-speed machining mode		0: Always incremental data command 1: Based on the modal G90/G91 command valid before high-speed machining mode is turned on
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
F84 (bit 6)	Tape run method		0: Not operated until the buffer is full. 1: Operated at a unit of EOB.
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F84 (bit 7)	Operation executed/not executed when the tool data has not bee set		When no tool data has been set during EIA/ISO program execution with the MAZATROL tool length data validated (parameter F93 bit 3 = 1): 0: Operation is executed. 1: Causes alarm 953 TOOL DATA INPUT PROCESS ERROR
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
F85	—		<p>Address Meaning Description</p> <p>7 6 5 4 3 2 1 0</p> <p>Table rotational machining 0: Always valid 1: Outside the area only Ignorance of radial interference check</p> <p>Type of coordinate system for controlling the tool tip point 0: The table coordinate system that rotates according to the particular rotation of the C-axis is defined as the programming coordinate system. 1: The work coordinate system is defined as the programming coordinate system.</p> <p>Tool tip point control scheme 0: Uniaxial rotation interpolation 1: Joint interpolation Fixed value (0)</p> <p>Reset to cancel G68.2 0: Valid 1: Invalid</p> <p>Display of surface definition 0 0: Invalid 1: Valid</p> <p>Output of the B-axis unclamping code before B-axis indexing 0: To output 1: Not to output (Valid only for MAZATROL programs of the INTEGREX and INTEGREX-e series.)</p> <ul style="list-style-type: none"> bit 3 = 0 <p>In the case of uni-axis rotational interpolation, the pivot point of the machine moves as denoted by the arrow. The tool attitude vector moves in a plane including R1 and R2.</p> bit 3 = 1 <p>In the case of joint interpolation, the pivot point of the machine moves as denoted by the arrow. The tool attitude vector moves along a conical surface.</p>
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

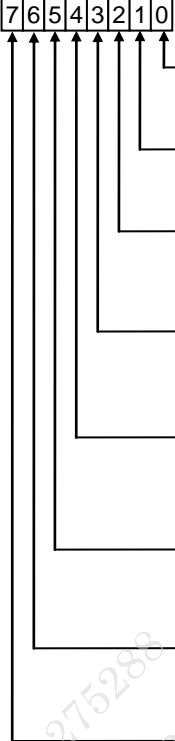
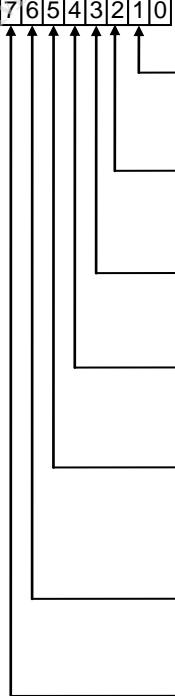
Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F86 (bit 0)	Output of M250 (Spindle Speed Confirmation) before a turning feed, valid/invalid		Output of M250 (Spindle Speed Confirmation) before a turning feed 0: Not output 1: Output
	Program type	M	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	
F86 (bit 1)	Milling-spindle start timing for a milling unit (with MILL&TURN. set under TYPE of UNo. 0)		Milling-spindle start timing for a milling unit (with MILL&TURN. set under TYPE of UNo. 0) 0: After tool change 1: At the initial point
	Program type	M	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	
F86 (bit 2)	Override scheme for G0 during tool tip point control		0: Makes the override function valid for the movement of the tool tip point.  1: Makes the override function valid for the clamping speed at the machine control point. If the moving speed of the axis exceeds the setting of machine parameter M1 (i.e., the maximum rapid feed rate), the override function will work for that setting. 
	Program type	E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F86 (bit 4)	Display of the PART SHAPE window		The plane for the display on the PART SHAPE window is: 0: X-Y plane 1: X-Z plane
	Program type	M	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	
F86 (bit 5)	Override scheme for G1 during tool tip point control		0: Override valid for the movement of the tool tip point 1: Override valid for the clamping speed at the machine control point
	Program type	E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	
F86 (bit 6)	Selection of rotary axis reference position for tool tip point control		0: Position during the start of tool tip point control 1: Position with the rotary axis at 0 degrees
	Program type	E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO								
Address	Meaning	Description									
F87	—	<p>Machine offset by each change in the deviation vector 0: Valid 1: Invalid</p> <p>Check for mismatch of the workpiece origin and table rotation center 0: Valid (an alarm is output in case of a mismatch) 1: Invalid</p> <p>Data alteration checking function (tool-related function) 0: Invalid 1: Valid</p> <p>Select a feed rate display method for the superposition system. 0: Display in terms of moving speed of actual axis 1: Display in terms of relative speed with respect to the workpiece</p> <p>Single-block mode cancellation during fixed cycle 0: Invalid (single-block stop occurs) 1: Valid (single-block stop does not occur)</p> <p>Common variables (#500 and on) are cleared by resetting or at the program end. 0: Invalid 1: Valid</p> <p>Local variables are cleared by resetting or at the program end. 0: Invalid 1: Valid</p> <p>Common variables (#100 to #199) are cleared by resetting or at the program end. 0: Invalid 1: Valid</p>									
	<p>Note : Variables for tool path check are not cleared even if Bit 5, 6 or 7 in F87 is set to 1.</p> <table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>At power on</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>Binary, eight digits</td> </tr> </table>			Program type	M, E	Conditions	At power on	Unit	Bit	Setting range	Binary, eight digits
Program type	M, E										
Conditions	At power on										
Unit	Bit										
Setting range	Binary, eight digits										

Classification	USER	Display title	EIA/ISO																																																																																																
F88			<p>Address</p> <p>Meaning</p> <p>Description</p> <p>Set this parameter to specify functions related to the conversion from MAZATROL program into an EIA program.</p> <table border="1"> <tr><td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td></tr> <tr><td colspan="4">Conversion of a part of program into sub-program (See Note.)</td><td colspan="4"></td></tr> <tr><td colspan="4">Output method of G-code for point machining</td><td colspan="4"></td></tr> <tr><td colspan="4">0: G-code of fixed cycle 1: G-code in 1 digit</td><td colspan="4"></td></tr> <tr><td colspan="4">Output of G10 and G92.5 in conversion of WPC data</td><td colspan="4"></td></tr> <tr><td colspan="4">0: Not to output 1: To output</td><td colspan="4"></td></tr> <tr><td colspan="4">Output of tool diameter correction G-code</td><td colspan="4"></td></tr> <tr><td colspan="4">0: Not to output G-code 1: To output G-code</td><td colspan="4"></td></tr> <tr><td colspan="4">Tool path modification caused by tool diameter correction</td><td colspan="4"></td></tr> <tr><td colspan="4">0: Correction is not included in path 1: Correction is included in path</td><td colspan="4"></td></tr> <tr><td colspan="4">Fixed value (0)</td><td colspan="4"></td></tr> <tr><td colspan="4">Fixed value (1)</td><td colspan="4"></td></tr> </table> <p>Note :</p> <p>When bit 0 of F88 is set to 1 on converting to the EIA program, the line machining, groove machining, face machining portions in the MAZATROL program are respectively programmed as subprograms.</p> <p>Example :</p> <p>Note :</p> <ul style="list-style-type: none"> 1. Subprograms that can be called out of the MAZATROL program are not reprogrammed as subprograms of the EIA program. 2. Manual program mode unit is not programmed as subprogram. 3. It can be selected to make a subprogram with parameter. <p>F88 bit 0 1: To make subprogram 0: Not to make subprogram</p> <p>Conversion into EIA program</p> <p>Main program</p> <pre>(WNo.1001); G17G0G40G54G64G90G94; G80; : [1] N1 (UNo.2FACE-MILL); M98P1001H1; G1Z-2; : M98P1001H1; G0Z100; [2] N2 (UNo.3LINE-CTR); M98P1001H2; G0Z-2; : N3 (UNo.4END); [3] : M30; %</pre> <p>Subprogram</p> <pre>N1 (UNo.2FACE-MILL); G0X5.Y2. ; G1X50. ; : [4] G0X60.Y60. ; M99; N2 (UNo.3LINE-CTR); G2X10.Y10.R5. ; G1X50. ; : [5] G2X40.Y10.R5. ; M99; %</pre> <p>Unit of machining units</p> <p>Unit of machining units</p> <p>Note : Bit 2 and 3 of F88 are valid only for machining centers.</p>	7	6	5	4	3	2	1	0	Conversion of a part of program into sub-program (See Note.)								Output method of G-code for point machining								0: G-code of fixed cycle 1: G-code in 1 digit								Output of G10 and G92.5 in conversion of WPC data								0: Not to output 1: To output								Output of tool diameter correction G-code								0: Not to output G-code 1: To output G-code								Tool path modification caused by tool diameter correction								0: Correction is not included in path 1: Correction is included in path								Fixed value (0)								Fixed value (1)							
7	6	5	4	3	2	1	0																																																																																												
Conversion of a part of program into sub-program (See Note.)																																																																																																			
Output method of G-code for point machining																																																																																																			
0: G-code of fixed cycle 1: G-code in 1 digit																																																																																																			
Output of G10 and G92.5 in conversion of WPC data																																																																																																			
0: Not to output 1: To output																																																																																																			
Output of tool diameter correction G-code																																																																																																			
0: Not to output G-code 1: To output G-code																																																																																																			
Tool path modification caused by tool diameter correction																																																																																																			
0: Correction is not included in path 1: Correction is included in path																																																																																																			
Fixed value (0)																																																																																																			
Fixed value (1)																																																																																																			
Program type		M, E																																																																																																	
Conditions		Immediate																																																																																																	
Unit		Bit																																																																																																	
Setting range		Binary, eight digits																																																																																																	

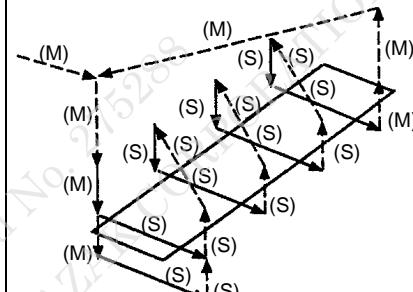
Classification	USER	Display title	EIA/ISO								
F89			<p>Set this parameter to specify functions related to the conversion from MAZATROL program into an EIA program.</p> <table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>Binary, eight digits</td> </tr> </table> <p>Description:</p> <ul style="list-style-type: none"> Output of shape data <ul style="list-style-type: none"> 0: Not to output 1: To output Synchronous tapping G-code <ul style="list-style-type: none"> 0: G74/G84 1: G84.2/G84.3 Output of F command <ul style="list-style-type: none"> 0: To output 1: Not to output Whether to generate an output for EIA conversion of the mill-turning unit <ul style="list-style-type: none"> 0: Alarm (No output) 1: Output of the path shifted through a distance equivalent to the tool radius EIA conversion output destination <ul style="list-style-type: none"> 0: Standard area 1: Backup area Returning to the second zero point before tool change (output of G30) <ul style="list-style-type: none"> 0: Not to return 1: To return <p>Fixed value (0)</p>	Program type	M, E	Conditions	Immediate	Unit	Bit	Setting range	Binary, eight digits
Program type	M, E										
Conditions	Immediate										
Unit	Bit										
Setting range	Binary, eight digits										
F91			<table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>At power on</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>Binary, eight digits</td> </tr> </table> <p>Description:</p> <ul style="list-style-type: none"> In response to move command without decimal point: <ul style="list-style-type: none"> 0: Tool moves by 1/1. 1: Tool moves by 10/1. Coordinate system shift using a MAZATROL program: <ul style="list-style-type: none"> 0: Invalid 1: Valid 0: Stroke inside check before movement <ul style="list-style-type: none"> 1: Stroke outside check before movement Metric/inch system (Note) <ul style="list-style-type: none"> 0: Metric (Initial G20 is valid/invalid) 1: Inch In response to move command without decimal point: <ul style="list-style-type: none"> 0: Tool moves in 0.0001 mm (0.00001 in) increments. 1: Tool moves in 1 mm (1 in) increments. 0: G00 interpolation <ul style="list-style-type: none"> 1: G00 non-interpolation 0: G33E command is for the number of threads per inch <ul style="list-style-type: none"> 1: G33E command is for thread cutting with precise lead <p>Note :</p> <p>Execute zero point return operation without fail before changing the setting of bit 4 (metric/inch system selection).</p>	Program type	M, E	Conditions	At power on	Unit	Bit	Setting range	Binary, eight digits
Program type	M, E										
Conditions	At power on										
Unit	Bit										
Setting range	Binary, eight digits										

Classification	USER	Display title	EIA/ISO								
F92	<p>—</p> <table border="1" style="margin-top: 10px;"> <tr> <td>Program type</td><td>M, E</td></tr> <tr> <td>Conditions</td><td>At power on</td></tr> <tr> <td>Unit</td><td>Bit</td></tr> <tr> <td>Setting range</td><td>Binary, eight digits</td></tr> </table>	Program type	M, E	Conditions	At power on	Unit	Bit	Setting range	Binary, eight digits		<p>Description</p> <ul style="list-style-type: none"> Modal at power-on or at reset (Initial G18) 0: G17 or G19 1: G18 Modal at power-on or at reset (Initial G19) 0: G17 or G18 1: G19 Dwell command method 0: Modal 1: Always based on the time command Tool-length compensation (G43, G44) axis for milling tools 0: X-, Y-, Z-axes (Note) 1: Z-axis fixed Tool-diameter compensation (G41 or G42) start up/cancel type 0: Type A 1: Type B Tool-diameter compensation (G41 or G42) interference check 0: Alarm stop occurs to prevent overcutting. 1: Tool path is changed to prevent overcutting. Fixed-cycle hole-drilling axis 0: Plane selection using G17, G18 or G19 1: Z-axis fixed ACT-φ/NOSE-R in the TOOL DATA display for an EIA/ISO program 0: Invalid 1: Valid <p>Note : In the case of tool offset type C, the value set for each axis on the TOOL OFFSET display is used for compensation.</p>
Program type	M, E										
Conditions	At power on										
Unit	Bit										
Setting range	Binary, eight digits										
F93	<p>—</p> <table border="1" style="margin-top: 10px;"> <tr> <td>Program type</td><td>M, E</td></tr> <tr> <td>Conditions</td><td>At power on</td></tr> <tr> <td>Unit</td><td>Bit</td></tr> <tr> <td>Setting range</td><td>Binary, eight digits</td></tr> </table>	Program type	M, E	Conditions	At power on	Unit	Bit	Setting range	Binary, eight digits		<ul style="list-style-type: none"> Modal at power-on or at reset 0: G94 (Feed per minute) 1: G95 (Feed per revolution) Modal at power-on or at reset 0: G91 (Incremental-value command) 1: G90 (Absolute-value command) Tool length of tool data for an EIA/ISO program 0: Invalid 1: Valid Feed rate during machine lock 0: Specified feed rate 1: Rapid feed rate Middle point during reference-point return 0: Return through middle point to reference point 1: Return directly to reference point Single-block operation mode at user macro operation instruction 0: Single-block stop does not occur (for operation). 1: Single-block stop occurs (for test). Fixed value (0)
Program type	M, E										
Conditions	At power on										
Unit	Bit										
Setting range	Binary, eight digits										

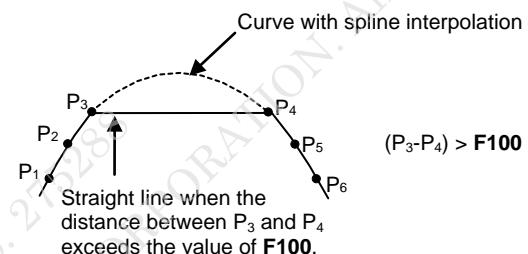
Classification	USER	Display title	EIA/ISO								
F94			<p>Address</p> <p>Meaning</p> <p>Description</p> <p>—</p> <table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>At power on</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>Binary, eight digits</td> </tr> </table>	Program type	M, E	Conditions	At power on	Unit	Bit	Setting range	Binary, eight digits
Program type	M, E										
Conditions	At power on										
Unit	Bit										
Setting range	Binary, eight digits										
F95 (bit 0)	Interrupt function using user macro instruction, valid/invalid		<p>Interrupt function using user macro instruction</p> <p>0: Invalid 1: Valid</p>								
F95 (bit 1)	Handling of macro program interruption and call		<p>Handling of macro program interruption and call</p> <p>0: Handled as interruption 1: Handled as subprogram call</p>								

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F95 (bit 2)	Automatic return position to restart the program (Fixed to 1)		Automatic return position to restart the program (Fixed to 1) 0: Manual return 1: Automatic return
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	
F95 (bit 3)	G00 (positioning) command feed rate for dry run		G00 (positioning) command feed rate for dry run 0: Rapid feed rate 1: Feed rate for dry run
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	
F95 (bit 4)	—		Invalid
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F95 (bit 5)	Alarm display when the slave or adaptor unit is not connected		Alarm display when the slave or adaptor unit is not connected 0: Displayed 1: Not displayed
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

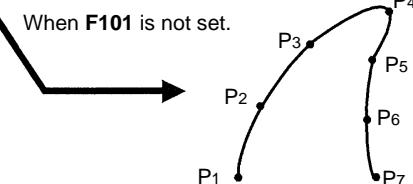
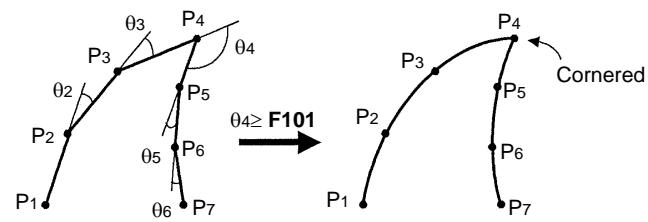
Classification	USER	Display title	EIA/ISO														
Address	Meaning	Description															
F95 (bit 6)	Manual-pulse interrupt amount cancellation with reset key, valid/invalid	Manual-pulse interrupt amount cancellation with reset key 0: Invalid 1: Valid															
	Program type	M, E															
	Conditions	At power on															
	Unit	—															
	Setting range	0, 1															
F95 (bit 7)	G54 coordinate system with the reset key, valid/invalid	With reset key 0: Coordinate system corresponding to G54 1: Coordinate system unchanged															
	Program type	M, E															
	Conditions	At power on															
	Unit	—															
	Setting range	0, 1															
F96		<table border="1"> <tr> <td>7</td> <td>6</td> <td>5</td> <td>4</td> <td>3</td> <td>2</td> <td>1</td> <td>0</td> </tr> <tr> <td>Selection of variable number for tool offset amount 0: 16001 to 16512, 17001 to 17512 1: 12001 to 12512, 13001 to 13512</td> <td>Fairing function 0: Invalid 1: Valid</td> <td>Processing for arc command blocks in high-speed machining mode 0: Nonuniform feed 1: Uniform feed</td> <td>Selection of a corner judgment criterion in high-speed machining mode 0: Judgment from the angle relative to adjacent blocks 1: Judgment by excluding the small block (if present between large-angle blocks)</td> <td>Selection of a cutting feed clamping speed in high-speed machining mode 0: Minimum clamping speed of movable axes 1: Clamping speed based on the radius of the curvature</td> <td>Rotational axis shape correction 0: Invalid 1: Valid</td> </tr> </table>	7	6	5	4	3	2	1	0	Selection of variable number for tool offset amount 0: 16001 to 16512, 17001 to 17512 1: 12001 to 12512, 13001 to 13512	Fairing function 0: Invalid 1: Valid	Processing for arc command blocks in high-speed machining mode 0: Nonuniform feed 1: Uniform feed	Selection of a corner judgment criterion in high-speed machining mode 0: Judgment from the angle relative to adjacent blocks 1: Judgment by excluding the small block (if present between large-angle blocks)	Selection of a cutting feed clamping speed in high-speed machining mode 0: Minimum clamping speed of movable axes 1: Clamping speed based on the radius of the curvature	Rotational axis shape correction 0: Invalid 1: Valid	
7	6	5	4	3	2	1	0										
Selection of variable number for tool offset amount 0: 16001 to 16512, 17001 to 17512 1: 12001 to 12512, 13001 to 13512	Fairing function 0: Invalid 1: Valid	Processing for arc command blocks in high-speed machining mode 0: Nonuniform feed 1: Uniform feed	Selection of a corner judgment criterion in high-speed machining mode 0: Judgment from the angle relative to adjacent blocks 1: Judgment by excluding the small block (if present between large-angle blocks)	Selection of a cutting feed clamping speed in high-speed machining mode 0: Minimum clamping speed of movable axes 1: Clamping speed based on the radius of the curvature	Rotational axis shape correction 0: Invalid 1: Valid												
	Program type	M, E															
	Conditions	At power on															
	Unit	Bit															
	Setting range	Binary, eight digits															

Classification	USER	Display title	EIA/ISO																			
Address	Meaning	Description																				
F97	Selection of G-code of the coordinates system to be used in the EIA conversion function		To select G-code modal of the coordinates system to be used in the EIA conversion function.																			
			<table border="1"> <thead> <tr> <th>Setting value</th> <th>Coordinates system</th> <th>Setting value</th> <th>Coordinates system</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>G54</td> <td>5</td> <td>G58</td> </tr> <tr> <td>2</td> <td>G55</td> <td>6</td> <td>G59</td> </tr> <tr> <td>3</td> <td>G56</td> <td></td> <td></td> </tr> <tr> <td>4</td> <td>G57</td> <td>Others</td> <td>G54</td> </tr> </tbody> </table>	Setting value	Coordinates system	Setting value	Coordinates system	1	G54	5	G58	2	G55	6	G59	3	G56			4	G57	Others
Setting value	Coordinates system	Setting value	Coordinates system																			
1	G54	5	G58																			
2	G55	6	G59																			
3	G56																					
4	G57	Others	G54																			
Program type	M, E																					
Conditions	Immediate																					
Unit	—																					
F98	Setting range	0 to 255																				
			<p>Note : F97 is valid only for machining centers.</p>																			
			<p>To specify the number of a macro variable to be used in the EIA conversion function. If any macro variable is not used, set to 0. In case of output with a subprogram in the EIA conversion, the height of cutting face is set with a macro variable. Set to F98 the number of the macro variable to be used.</p>																			
			 <p>(M) Main program (S) Subprogram ---- Rapid feed ← Cutting feed</p> <p>MPL504</p>																			
	Number of macro variable to be used in the EIA conversion function		<p>Main program</p> <pre>G01Z_ ; : To the height of # = ; Height of cutting face Macro variable (F98) M98P_H_ ; : M30; %</pre> <p>Subprogram</p> <pre>N_(_) ; G00Z_ ; : Return on Z-axis X_Y_ ; : Moves to the start point of the next machining on Z-axis XY_ ; : Machining on Z-axis specified with the variable M99; %</pre> <p>[Units that use macro variables] FCE MILL (cutting in one direction), TOP EMIL, POCKET, PCKT MT, PCKT VLY</p> <p>Note :</p> <ol style="list-style-type: none"> 1. 3D machining cannot be output using subprograms. 2. Subprogram is output in the absolute mode (G90). 																			
	Program type	M, E																				
	Conditions	Immediate																				
	Unit	—																				
	Setting range	100 to 199 500 to 999																				

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F99	Offset amount for the subprogram WNo. to the main WNo. concerned in case of output with subprogram in the EIA conversion function		In case of output with subprogram in the EIA conversion function: Example : WNo. 10 When F99 is "20". EIA conversion (The WNo. of the converted program is assumed to be 1000.) Main WNo. 1000 Sub WNo. 1020 Offset amount: 20 For the subprogram of the EIA conversion function, refer to F88 bit 0.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	1 to 99999998	
F100	Spline cancel length	If the commanded distance in a block exceeds the spline cancel length (F100), spline interpolation is not realized in this block even in the spline interpolation mode.	
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 in	
	Setting range	0 to 999999	
F101	Spline cancel angle	If the angle formed by two blocks exceeds the value set by the parameter F101 , spline interpolation is not realized in these blocks even in the spline interpolation mode. If F101 = 0 in the spline interpolation mode, the spline interpolation mode is not canceled. In the 5-axis spline interpolation mode, this sets the spline cancel angle at 60 degrees.	
	Program type	E	
	Conditions	Immediate	
	Unit	°	
	Setting range	0 to 179	

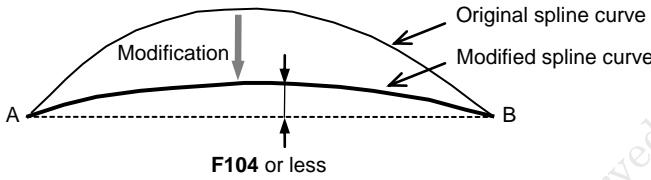
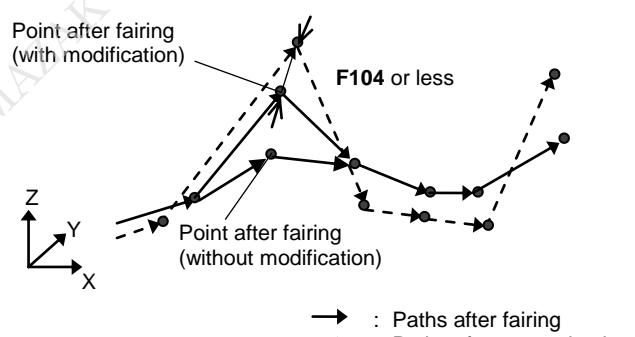
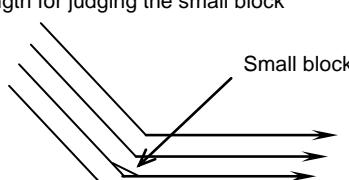


MPL505



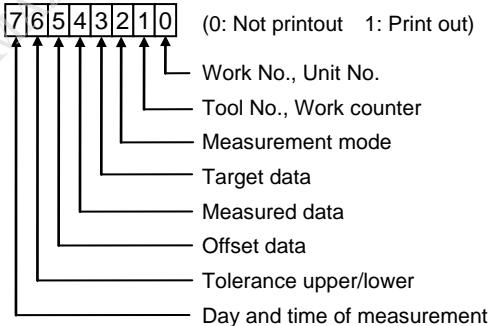
MPL506

Classification	USER	Display title	EIA/ISO
Address		Meaning	
F102		Fine spline interpolation curve error (Block including the point of inflection)	
		Program type	E
		Conditions	Immediate
		Unit	0.0001 mm/0.00001 in
		Setting range	0 to 99999999
F103		Spline interpolation fairing block length, 5-axis spline interpolation small block length	
		Program type	E
		Conditions	Immediate
		Unit	0.0001 mm/0.00001 in
		Setting range	0 to 99999999
		<p>During block checking in the fine spline interpolation mode, if the spline curve of a specific block is judged to include an inflection point and the maximum chord error between the spline curve and the block is larger than the value of F102, the shape of the curve will be modified to reduce the maximum chord error below the value of F102.</p> <p style="text-align: right;">D735S0001</p>	
		<p>If a block whose length is less than the value of F103 is detected during fine spline interpolation, that block will be skipped and integrated (faired) into the preceding and succeeding blocks to create a spline curve.</p> <p>Suppose that the i-th block in the fine spline interpolation mode has a block length of l_i:</p> <ul style="list-style-type: none"> If l_{i-1} is greater than $F103 \times 2$ l_i is equal to or less than F103 l_{i+1} is greater than $F103 \times 2$ <p>then the ending point of the "i - 1" th block and the starting point of the "i + 1" th block will be modified to the middle point of the i-th block and this block will be deleted. A spline curve will be created from the sequence of points updated this way.</p> <p style="text-align: right;">D735S0002</p>	
		<p>If the length of the starting block or ending block in the fine spline interpolation mode is smaller than the value of F103, processing will slightly differ from that described above.</p> <p>This parameter is effective when bit 1 of F96 is 1.</p>	
		<p>If the distance between blocks is less than the value of F103 in the 5-axis spline interpolation mode, the 5-axis spline interpolation mode will be canceled only between these blocks. When F103 is set to 0, the value of parameter F104 is used.</p>	

Classification	USER	Display title	EIA/ISO
F104	<p>Fine spline interpolation curve error (Block including no inflection point)</p> <p>5-axis spline interpolation curve error</p> <p>5-axis fairing command error</p>	<p>During block checking in the fine spline interpolation mode, if the spline curve of a specific block is judged to include no inflection point and the maximum chord error between the spline curve and the block is larger than the value of F104, the shape of the curve will be modified to reduce the maximum chord error below the value of F104.</p>  <p>D735S0003</p> <p>During block checking in the 5-axis spline interpolation mode, if the maximum chord error between the spline curve and the block is larger than the value of F104, the shape of the curve will be modified to reduce the maximum chord error below the value of F104.</p> <p>When F104 is set to 0, if the block is judged to include an inflection point, the spline curve of the block will be modified to be linear, and if it is judged to include no inflection point, it will not be modified (spline interpolation is realized).</p> <p>In 5-axis fairing, if the error between the command point and the point after fairing for the tool tip 3 axes is larger the value of F104, the point after fairing is modified to reduce the error below the value of F104.</p> <p>When F104 is set to 0, the point after fairing will not be modified.</p> 	
F107	<p>Small block judgment length</p>	<p>In a small-segment machining program, if a small block is present between large-angle blocks, optimum corner deceleration can be achieved by excluding the small block and then judging the total corner angle.</p> <p>Judgment length for judging the small block</p>  <p>This parameter is valid when bit 4 of F96 is 1.</p>	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F108	Corner deceleration angle increment value		Specify by what degrees the angle of corner deceleration is to be incremented when the shape correction function is valid.
	Program type	M, E	
	Conditions	Immediate	
	Unit	°	
	Setting range	0 to 99999999	
F109	Number of common variables between turrets (#100 onward)		Set the number of common variables which can be read and written between the turrets. Example : If 10 is set in F109 #100 to #109: Common variables that are valid for both turrets #110 to #199: Common variables that are valid only for one turret
	Program type	E	
	Conditions	Immediate	
	Unit	Number of variables	
	Setting range	0 to 100	
F110	Number of common variables between turrets (#500 onward)		Set the number of common variables which can be read and written between the turrets. Example : If 10 is set in F110 #500 to #509: Common variables that are valid for both turrets #510 to #999: Common variables that are valid only for one turret
	Program type	E	
	Conditions	Immediate	
	Unit	Number of variables	
	Setting range	0 to 500	
F111 (bit 0)	Selection of display type of tapping tool on the VIRTUAL MACHINING or 3D MONITOR display		On the VIRTUAL MACHINING or 3D MONITOR display: F111 (bit 0) = 0: Simplified display of tapping tool F111 (bit 0) = 1: Detailed display of tapping tool
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F111 (bit 1)	Use/disuse of dry run during thread cutting		<p>F111 (bit 1) = 0: Disuse of dry run during thread cutting cycle</p> <p>F111 (bit 1) = 1: Use of dry run during thread cutting cycle</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F111 (bit 2)	Use/disuse of feed hold during thread cutting		<p>F111 (bit 2) = 0: The feed hold function causes the threading operation not to be stopped till completion of the next step.</p> <p>F111 (bit 2) = 1: The feed hold function causes the threading operation to be changed into run-out and then stopped.</p> <p>Note : With TC82 = 0, the same operation as that for setting 0 will occur even if 1 is set for F111 bit 2.</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F111 (bit 3)	Direction of rotation of the C-axis during C-axial threading with G01.1		<p>Select the direction of rotation of the C-axis during C-axial threading based on G01.1.</p> <p>F111 (bit 3) = 0: The C-axis rotates CW (forward).</p> <p>F111 (bit 3) = 1: The C-axis rotates CCW (backward).</p>
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F111 (bit 5)	Tool correction amount selection for EIA/ISO programs		<p>Select whether the wear correction data on the TOOL DATA display is to be added during execution of the EIA/ISO program when using the tool length that has been entered on the TOOL DATA display.</p> <p>F111 (bit 5) = 0: Wear correction data is not added</p> <p>F111 (bit 5) = 1: Wear correction data is added</p>
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F111 (bit 6)	Execution mode selection for a fixed turning cycle		In the fixed turning cycle mode (G290/G292/G294) F111 (bit 6) = 0: The fixed turning cycle is executed for each block (G66 type). F111 (bit 6) = 1: The fixed turning cycle is executed only for movement blocks (G66.1 type).
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F111 (bit 7)	Form of single-block stop during a fixed turning cycle		This parameter specifies whether single-block operation during a fixed turning cycle (G290, G292 or G294) is to be stopped after the entire cycle has been executed, or for each block. F111 (bit 7) = 0: After execution of the cycle F111 (bit 7) = 1: For each block
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F112	Selection of measurement data items to be printed out		Selection of printout items in measurement data printout 
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
F113 (bit 0)	Counting all types of use under the same tool number for the tool life management on the TOOL DATA display executed/not executed		F113 (bit 0) = 0: Counting each type of use under the same tool number individually. F113 (bit 0) = 1: Counting all types of use under the same tool number integrally.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F113 (bit 1)	Data handling on the milling tool of a group that has expired in tool life Program type M, E Conditions Immediate Unit — Setting range 0, 1	F113 (bit 1) = 0: Operation will be continued. F113 (bit 1) = 1: Operation will be stopped.	
F113 (bit 2)	Data handling on the turning tool of a group that has expired in tool life Program type M, E Conditions Immediate Unit — Setting range 0, 1	F113 (bit 2) = 0: Operation will be continued. F113 (bit 2) = 1: Operation will be stopped.	
F113 (bit 3)	Tool life management of the FLASH tool Program type M, E Conditions Immediate Unit — Setting range 0, 1	Select whether the FLASH tool that has been registered under the same tool number for the tool life management function is to be included in spare tools. F113 (bit 3) = 0: The FLASH tool will be included in spare tools. F113 (bit 3) = 1: The FLASH tool will not be included in spare tools.	
F113 (bit 4)	Tool life management – Life time Program type M, E Conditions Immediate Unit — Setting range 0, 1	Select whether the life time on the TOOL DATA display is to be included in the life judgment items listed for the tool life management function. F113 (bit 4) = 0: The life time will be included in the life judgment items. F113 (bit 4) = 1: The life time will not be included in the life judgment items.	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F113 (bit 5)	Tool life management – Maximum available wear offset data X		Select whether the maximum available wear offset data X on the TOOL DATA display is to be included in the life judgment items listed for the tool life management function. F113 (bit 5) = 0: The maximum available wear offset data X will be included in the life judgment items. F113 (bit 5) = 1: The maximum available wear offset data X will not be included in the life judgment items.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F113 (bit 6)	Tool life management – Maximum available wear offset data Y		Select whether the maximum available wear offset data Y on the TOOL DATA display is to be included in the life judgment items listed for the tool life management function. F113 (bit 6) = 0: The maximum available wear offset data Y will be included in the life judgment items. F113 (bit 6) = 1: The maximum available wear offset data Y will not be included in the life judgment items.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F113 (bit 7)	Tool life management – Maximum available wear offset data Z		Select whether the maximum available wear offset data Z on the TOOL DATA display is to be included in the life judgment items listed for the tool life management function. F113 (bit 7) = 0: The maximum available wear offset data Z will be included in the life judgment items. F113 (bit 7) = 1: The maximum available wear offset data Z will not be included in the life judgment items.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F114 (bit 0)	Selection of the maximum C-axial cutting feed rate for the inch system		Specify the maximum C-axial cutting feed rate that can be selected for the inch system. F114 (bit 0) = 0: The maximum selectable C-axial cutting feed rate is 88 min^{-1} . F114 (bit 0) = 1: The maximum selectable C-axial cutting feed rate is 400 min^{-1} .
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

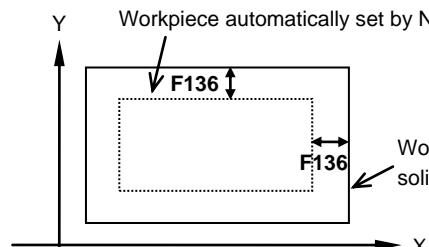
Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F114 (bit 1)	Selection of the operation occurring during the control of the tool tip point when command G49 is issued (when the tool length offset value is canceled) Program type M, E Conditions Immediate Unit — Setting range 0, 1	Select the type of operation occurring during the control of the tool tip point when command G49 is issued (when the tool length offset value is canceled). F114 (bit 1) = 0: The axis moves according to the tool length offset value. F114 (bit 1) = 1: The axis does not move. .	
F114 (bit 2)	B-axis angle check during measurement of the turning tool for which OUT is specified in "section to be machined" Program type M, E Conditions At power on Unit — Setting range 0, 1	Select whether the B-axis angle is to be checked during measurement of the turning tool for which OUT is specified in "section to be machined". F114 (bit 2) = 0: During measurement for OUT turning tool, alarm 653 ILLEGAL TOOL DESIGNATED occurs if the B-axis angle is not 90°. F114 (bit 2) = 1: The measurements for OUT turning tool are possible irrespective of the B-axis angle. Turning tools for which IN or EDGE is specified in "section to be machined" can be measured irrespective of the B-axis angle.	
F114 (bit 3)	Moving axes by using G49 (tool length offset cancel) in G43 (tool length offset) mode Program type E Conditions At power on Unit — Setting range 0, 1	Moving axes by using G49 (tool length offset cancel) in G43 (tool length offset) mode F114 (bit 3) = 0: Valid F114 (bit 3) = 1: Invalid	
F114 (bit 4)	Retreading function Program type M, E Conditions Immediate Unit — Setting range 0, 1	Selecting a retreading function F114 (bit 4) = 0: Using the retreading function F114 (bit 4) = 1: Not using the retreading function	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F114 (bit 5)	Output timing of a tool life alarm		Output timing of a tool life alarm F114 (bit 5) = 0: At the time of the next tool change F114 (bit 5) = 1: When the program ends.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F114 (bit 6)	Initial G53.5		F114 (bit 6) = 0: Invalid. Initially, the G50-specified coordinate system is selected. F114 (bit 6) = 1: Valid. Initially, the MAZATROL coordinate system (G53.5) is selected.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F114 (bit 7)	Judging tool life from the count of machined workpieces		F114 (bit 7) = 0: The life of the tool is judged from its machining count. F114 (bit 7) = 1: The life of the tool is not judged from its machining count.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F115	Restart/TPS approach speed		Set the approach speed existing before cutting feed is started in restart/TPS mode.
	Program type	M	
	Conditions	Immediate	
	Unit	mm/min / 0.1 in/min	
	Setting range	0 to 65535	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F116	Feed rate of the threading runout – X-axis	<p>Specify the X-axis feed rate for the runout of the threading cycle.</p> <p>A quarter (1/4) of the rapid traverse (as set in parameter M1) applies when this parameter is set to zero (0).</p> <p>In case of M1 < F116, the feed rate will be clamped with the M1 value.</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	mm/min	
	Setting range	0 to 240000	
F117	Feed rate of the threading runout – Y-axis	<p>Specify the Y-axis feed rate for the runout of the threading cycle.</p> <p>A quarter (1/4) of the rapid traverse (as set in parameter M1) applies when this parameter is set to zero (0).</p> <p>In case of M1 < F117, the feed rate will be clamped with the M1 value.</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	mm/min	
	Setting range	0 to 240000	
F118	Feed rate of the threading runout – Z-axis	<p>Specify the Z-axis feed rate for the runout of the threading cycle.</p> <p>A quarter (1/4) of the rapid traverse (as set in parameter M1) applies when this parameter is set to zero (0).</p> <p>In case of M1 < F118, the feed rate will be clamped with the M1 value.</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	mm/min	
	Setting range	0 to 240000	
F119	Runout feed rate for the inside diameter threading cycle	<p>Specify the runout feed rate for the inside diameter threading cycle.</p> <p>This parameter is valid only when chamfering is not specified in the inside diameter threading unit of the MAZATROL program.</p> <p>A quarter (1/4) of the rapid traverse (as set in parameter M1) applies when this parameter is set to zero (0).</p> <p>In case of M1 < F119, the feed rate will be clamped with the M1 value.</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	mm/min	
	Setting range	0 to 240000	

Classification	USER	Display title	EIA/ISO	
Address	Meaning	Description		
F120	Clamping speed for the threading cycle – X-axis		Specify the X-axis clamping speed for the threading cycle.	
	Program type	M, E		
	Conditions	Immediate		
	Unit	mm/min		
	Setting range	0 to 99999999		
F121	Clamping speed for the threading cycle – Y-axis		Specify the Y-axis clamping speed for the threading cycle.	
	Program type	M, E		
	Conditions	Immediate		
	Unit	mm/min		
	Setting range	0 to 99999999		
F122	Clamping speed for the threading cycle -- Z-axis		Specify the Z-axis clamping speed for the threading cycle.	
	Program type	M, E		
	Conditions	Immediate		
	Unit	mm/min		
	Setting range	0 to 99999999		
F124	Permissible data alteration amount 1 for input error prevention function		The data input in the following items are checked based on the permissible data alteration amount set in this parameter. TOOL DATA: LENGTH, ACT-φ, TOOL SET X, TOOL SET Z, LENG COMP., LENG.CO., ACT-φCO. TOOL OFFSET: GEOMETRIC OFFSET	
	Program type	M, E		
	Conditions	Immediate		
	Unit	0.0001 mm/0.00001 in		
	Setting range	0 to 99999999		

Classification	USER		Display title	EIA/ISO
Address	Meaning		Description	
F125	Permissible data alteration amount 2 for input error prevention function		<p>The data input in the following items are checked based on the permissible data alteration amount set in this parameter.</p> <p>TOOL DATA: WEAR COMP. X, WEAR COMP. Y, WEAR COMP. Z</p> <p>TOOL OFFSET: WEAR COMP.</p>	
	Program type	M, E		
	Conditions	Immediate		
	Unit	0.0001 mm/0.00001 in		
	Setting range	0 to 99999999		
F133	Pitch of tapping tool for display in detail on the VIRTUAL MACHINING or 3D MONITOR display		<p>Set the pitch of tapping tool displayed when the tapping tool is displayed in detail (F111 bit 0 = 1) in solid mode on the VIRTUAL MACHINING or 3D MONITOR display.</p>	
	Program type	M		
	Conditions	Immediate		
	Unit	0.1 mm/0.01 in		
	Setting range	0 to 65535		
F134	Thread depth of tapping tool for display in detail on the VIRTUAL MACHINING or 3D MONITOR display		<p>Set the thread depth of tapping tool displayed when the tapping tool is displayed in detail (F111 bit 0 = 1) in solid mode on the VIRTUAL MACHINING or 3D MONITOR display</p>	
	Program type	M		
	Conditions	Immediate		
	Unit	0.1 mm/0.01 in		
	Setting range	0 to 65535		
F135	Tool-drawing accuracy on the VIRTUAL MACHINING or 3D MONITOR display		<p>Specify tool-drawing accuracy on the VIRTUAL MACHINING or 3D MONITOR display</p> <p>As the specified value is greater (maximum value: 9), accuracy increases progressively.</p>	
	Program type	M		
	Conditions	Immediate		
	Unit	—		
	Setting range	0 to 9		

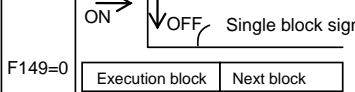
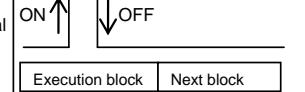
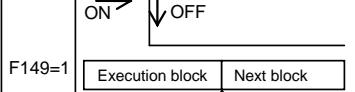
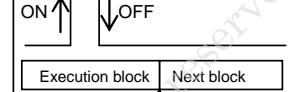
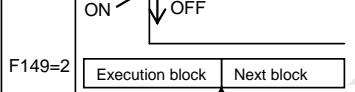
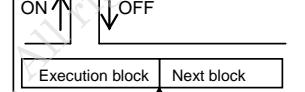
Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F136	Amount of offset for dummy workpiece shape on the VIRTUAL MACHINING or 3D MONITOR display		It is possible to change the size of the workpiece created automatically on the VIRTUAL MACHINING or 3D MONITOR display.
	Program type	M	 <p>Workpiece automatically set by NC</p> <p>Workpiece displayed in solid display mode</p> <p>This offset function is valid for both X-axial and Y-axial directions.</p>
	Conditions	Immediate	
	Unit	0.1 mm/0.01 in	
	Setting range	-99999999 to 99999999	
F137	Number of jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for No. 1 turning spindle		Specify the number of jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display automatically. If 0 is entered, three jaws will be displayed as standard.
	Program type	M	<p>Number of jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for No. 1 turning spindle</p>
	Conditions	Immediate	
	Unit	Jaws	
	Setting range	0 to 9	
F138	Number of jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for No. 2 turning spindle		Specify the number of jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display automatically. If 0 is entered, three jaws will be displayed as standard.
	Program type	M	<p>Number of jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for No. 2 turning spindle</p>
	Conditions	Immediate	
	Unit	Jaws	
	Setting range	0 to 9	
F139	Angle offset for the jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for the No. 1 turning spindle		Specify the amount of C-axial angle offset for the jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display automatically.
	Program type	M	<p>Angle offset for the jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for the No. 1 turning spindle</p>
	Conditions	Immediate	
	Unit	0.1°	
	Setting range	0 to 3599	

Classification	USER	Display title	EIA/ISO
F140	Angle offset for the jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display for the No. 2 turning spindle	Program type Conditions Unit Setting range	Specify the amount of C-axial angle offset for the jaws indicated on the VIRTUAL MACHINING or 3D MONITOR display automatically. M Immediate 0.1° 0 to 3599
F141	Roughness level of the circular milling unit (precision tornado cycle)	Program type Conditions Unit Setting range	Specify the roughness level of bottom when the precision rapid boring tornado cycle (created by M640) is program-converted into the MATRIX tornado milling cycle. M, E Immediate — 0 to 9
F143 (bit 0)	Operation when workpiece counting is made valid while the automatic operation INTELLIGENT SAFETY SHIELD (ISS) is enabled	Program type Conditions Unit Setting range	Operation when workpiece counting is made valid while the automatic operation INTELLIGENT SAFETY SHIELD (ISS) is enabled. <If workpiece counting is selected manually> 0: Workpiece counting is activated while the ISS is disabled, and the automatic operation continues. 1: Workpiece counting is not activated, the alarm 1129 MEMORY PROTECT (SAFETY SHIELD) is displayed and the automatic operation continues. <If workpiece counting is specified in the program> 0: Workpiece counting is activated while the ISS is disabled, and the automatic operation continues. 1: Workpiece counting is not activated, the alarm 1824 UNWRITABLE SYSTEM VARIABLE (ISS) is displayed and the automatic operation stops.
F143 (bit 1)	Status of the [STORE WORKPCE MATERIAL] menu function when the power is turned on	Program type Conditions Unit Setting range	Select the status of the [STORE WORKPCE MATERIAL] menu function when the power is turned on. 0: Disabled (The condition of the material model cut with the ISS enabled is not maintained.) 1: Enabled (The condition of the material model cut with the ISS enabled is maintained.)

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F143 (bit 2)	Operation when a specific G-code is issued while the automatic operation INTELLIGENT SAFETY SHIELD is enabled		<p>Operation when a specific G-code is issued while the automatic operation INTELLIGENT SAFETY SHIELD (ISS) is enabled.</p> <p>0: The automatic operation continues with the ISS left enabled.</p> <p>1: The alarm 1127 INVALID SAFETY SHIELD is displayed, the ISS is disabled and the automatic operation continues.</p>
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	
F143 (bit 6)	Display of WPC during inclined-surface machining on the POSITION display		<p>Display of WPC on the POSITION display during inclined-surface machining</p> <p>0: Display the work coordinate system that is selected before the inclined-surface machining command.</p> <p>1: Display the coordinate system that is selected after coordinate system rotation and zero point parallel movement according to the inclined-surface machining command.</p>
	Program type	E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F143 (bit 7)	G43 modal cancel selection with the RESET key		<p>Cancel or hold G43 when the reset key is pressed in G43 (tool length compensation) mode.</p> <p>0: Cancel</p> <p>1: Hold</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F144 (bit 1)	Selection of table rotary axis reference position for inclined-surface machining		<p>Selection of table rotary axis reference position for inclined-surface machining</p> <p>0: Table rotary axis start position as the reference</p> <p>1: Table rotary axis 0-degree position as the reference</p>
	Program type	E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F144 (bit 3)	Setting a minus value for argument Q in hobbing, valid/invalid		<p>Setting a minus value for argument Q (module/diametral pitch) in hobbing</p> <p>0: Invalid 1: Valid</p> <p>* If a minus value is set for argument Q in hobbing, while 0 is set for F144 bit 3, alarm 809 ILLEGAL NUMBER INPUT will be caused.</p>
	Program type	E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F144 (bit 5)	Selection of operation by the G49 command immediately after a macro variable command		<p>G49 command immediately after the macro variable command when bit 3 of parameter F114 is set to 1 (no axis movement by the G49 command)</p> <p>0: No axis movement 1: Axis movement</p>
	Program type	E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F144 (bit 6)	Deletion of unnecessary codes during EIA/ISO conversion for machining centers		<p>G92.5 block, #3090 = 1, and G10.9 block in the EIA/ISO program converted for machining centers (parameter F30 is set to 0)</p> <p>0: Output 1: Do not output</p> <p>Note : For G92.5 block only, the setting at bit 7 of parameter F144 is used.</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F144 (bit 7)	Deletion of G92.5 blocks during EIA/ISO conversion for machining centers		<p>G92.5 block in the EIA/ISO program converted for machining centers (parameter F30 is set to 0)</p> <p>0: Output 1: Do not output</p> <p>* Same as when bit 6 of parameter F144 is set to 1</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	

Classification	USER		Display title	EIA/ISO
Address	Meaning		Description	
F145	Rapid feed override when data alteration is detected		<p>The rapid feed rate override is reduced to the value set here, when data alteration is detected.</p> <p>This value will be handled as 100 if 0 is set here.</p>	
	Program type	M		
	Conditions	Immediate		
	Unit	%		
	Setting range	0 to 100		
F146	Selection of the material model's part to be taken off (hidden) on the VIRTUAL MACHINING and 3D MONITOR displays in the flow of turning operation under synchronization of both spindles		<p>Representation of cutting-off or parting operation under synchronization of both turning spindles on the VIRTUAL MACHINING and 3D MONITOR displays</p> <p>0: No representation (Cut-off part remains shown in its original place and undergoes the interference check.)</p> <p>1: Representation by taking-off the material from the 2nd spindle, with the material remaining on the 1st spindle (Cut-off part is hidden and excluded from the interference check.)</p> <p>2: Representation by taking-off the material from the 1st spindle, with the material remaining on the 2nd spindle (Cut-off part is hidden and excluded from the interference check.)</p>	
	Program type	M, E		
	Conditions	Immediate		
	Unit	—		
	Setting range	0 to 2		
F147	Selection of the material model's part to be taken off (hidden) on the VIRTUAL MACHINING and 3D MONITOR displays in the flow of milling operation		<p>Representation of cutting-off or parting milling operation with the workpiece gripped by both chucks on the VIRTUAL MACHINING and 3D MONITOR displays</p> <p>0: No representation (Cut-off part remains shown in its original place and undergoes the interference check.)</p> <p>1: Representation by taking-off the material from the 2nd spindle, with the material remaining on the 1st spindle (Cut-off part is hidden and excluded from the interference check.)</p> <p>2: Representation by taking-off the material from the 1st spindle, with the material remaining on the 2nd spindle (Cut-off part is hidden and excluded from the interference check.)</p>	
	Program type	M, E		
	Conditions	Immediate		
	Unit	—		
	Setting range	0 to 2		
F148	Compensation amount for G41/G42 given in a manual program (MANL PRG) unit		<p>Reference data for the calculation of compensation amount for G41/G42 given in a manual program unit</p> <p>F148 = 0: NOSE-R setting on the TOOL OFFSET display</p> <p>F148 = 1: NOSE-R or ACT-φ setting on the TOOL DATA display</p>	
	Program type	M		
	Conditions	Immediate		
	Unit	—		
	Setting range	0, 1		

Classification	USER	Display title	EIA/ISO								
Address	Meaning	Description									
F149	Operation between blocks when single block mode is switched from ON to OFF during automatic operation		Operation between blocks in the single block signal status when single block mode is switched from ON to OFF during automatic operation								
			When the single block signal is turned from ON to OFF When the single block signal is turned from OFF to ON and then to OFF								
			 								
			 								
			 								
	Program type	M, E									
	Conditions	Immediate									
	Unit	—									
	Setting range	0 to 2									
	Output data offset of NC basic information for field network		Set the top register R (R18176 to R18463) for output destination of NC basic information for field network.								
F150			<p>Example :</p> <p>1. When F150 = 0 2. When F150 = 352</p> <table border="1" style="margin-left: auto; margin-right: auto;"> <tr> <td style="text-align: center;">R18176 ⋮ R18463 R18464 ⋮ R18815</td> <td style="text-align: center;">Output side NC basic information Data output to slave/adaptor</td> <td style="text-align: center;">R18176 ⋮ R18527 R18528 ⋮ R18815</td> <td style="text-align: center;">Output side Data output to slave/adaptor NC basic information</td> </tr> </table>	R18176 ⋮ R18463 R18464 ⋮ R18815	Output side NC basic information Data output to slave/adaptor	R18176 ⋮ R18527 R18528 ⋮ R18815	Output side Data output to slave/adaptor NC basic information				
R18176 ⋮ R18463 R18464 ⋮ R18815	Output side NC basic information Data output to slave/adaptor	R18176 ⋮ R18527 R18528 ⋮ R18815	Output side Data output to slave/adaptor NC basic information								
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0 to 352										
		<p>Note :</p> <p>If a value set for F150 is out of the setting range or is an odd value, alarm 528 FIELDNETWORK SETTING ERROR (4, ,) occurs and the communication stops. NC basic information will not be output.</p>									
		Data to be referenced or changed by macro variable #3001 (power-on time)									
		0: Value indicated for POWER ON in the ACCUMULATED TIME window									
		1: Value counted from the power-on timing									
		<table border="1" style="width: 100%;"> <tr> <td style="width: 30%;">F151 bit7</td> <td style="width: 40%;">Value indicated for POWER ON in the ACCUMULATED TIME window at the time when the read command to #3001 is specified</td> <td style="width: 30%;">Value at #3001 at the time the power is turned on again</td> </tr> <tr> <td>0</td> <td>Changed accordingly</td> <td>Not reset</td> </tr> <tr> <td>1</td> <td>Not changed</td> <td>Reset to 0</td> </tr> </table>	F151 bit7	Value indicated for POWER ON in the ACCUMULATED TIME window at the time when the read command to #3001 is specified	Value at #3001 at the time the power is turned on again	0	Changed accordingly	Not reset	1	Not changed	Reset to 0
F151 bit7	Value indicated for POWER ON in the ACCUMULATED TIME window at the time when the read command to #3001 is specified	Value at #3001 at the time the power is turned on again									
0	Changed accordingly	Not reset									
1	Not changed	Reset to 0									
F151 (bit 7)	Program type	E									
	Conditions	Immediate									
	Unit	—									
	Setting range	0, 1									
			[Related macro variable] #3040 : Power-on time in the ACCUMULATED TIME window (read only)								

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F152	—		Fixed value (0)
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F153	SMART font display, valid/invalid		<p>Make SMART font display valid/invalid. 0: Invalid 1: Valid</p> <p>* This parameter is valid only when any language other than Japanese, Chinese (traditional/simplified), Korean, Russian or Bulgarian is selected for the display language.</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F154	—		<p>Parameter for system internal setting Setting prohibited</p>
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F155 (bit 3)	Operation when argument I is omitted from the tap cycle command having argument Q		<p>Operation when argument I is omitted from the tap cycle command having argument Q 0: Operation as a deep-hole cycle 1: Operation as a high-speed deep-hole cycle</p>
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	

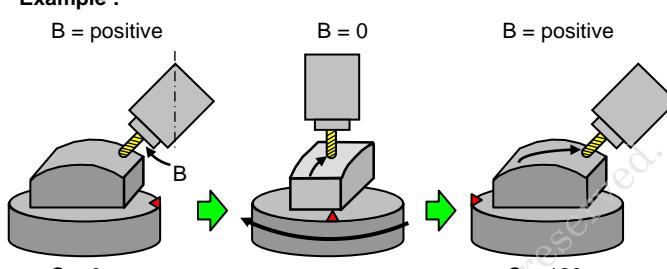
Classification	USER	Display title	EIA/ISO
F156 (bit 2)	Separation between rotary axis and linear axis when moving axes during workpiece placement error correction	Program type Conditions Unit Setting range	Description Separation between rotary axis and linear axis when moving axes during workpiece placement error correction 0: Combine the movement of rotary axis and linear axis. 1 : Separate the movement between rotary axis and linear axis.
F156 (bit 3)	Selection of solution to workpiece placement error correction (primary rotary axis sign prioritized)	Program type Conditions Unit Setting range	Selection of solution to workpiece placement error correction (primary rotary axis sign prioritized) 0 : No priority according to the sign in the command (compliant with F162 bit 1) 1 : Priority according to the sign in the command in the program
F156 (bit 4)	Pitch division in the case when the arc length exceeds the maximum value during helical/spiral interpolation	Program type Conditions Unit Setting range	Pitch division in the case when the arc length exceeds the maximum value during helical/spiral interpolation 0: Invalid 1: Valid Remark : Arc length [mm] = diameter [mm] × π × pitch [times] Maximum arc length: 107374.1824 mm * When this parameter is invalid, alarm 814 INTERPOLATION OVERFLOW will occur if the arc length has exceeded the maximum length.
F156 (bit 5)	Single-block stop when the VFC key is pressed with override set to 100%	Program type Conditions Unit Setting range	Set a stop at the end of execution of a tool sequence block when the VFC key is pressed with override set to 100% during automatic operation. 0: Single-block stop does not occur 1: Single-block stop occurs

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F156 (bit 6)	—	Fixed value (0)	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F157 (bit 2)	EIA tip position memory	Tip position memory setting in consideration of the tool length compensation in the EIA tip position memory window on the WORK OFFSET display when the MAZATROL tool length is not valid 0: Invalid 1: Valid * This parameter is valid when parameter F30 is set to 0 and bit 3 of parameter F93 is set to 0 (MAZATROL tool length invalid).	
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
F157 (bit 5)	Add-in EIA function	Add-in EIA function 0: Invalid 1: Valid	
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
F158 (bit 0)	Tool length/tool diameter check function after tool change	This function stops operation with alarm 193 NO TOOL IN MAGAZINE if the tool length or tool diameter value is not set when this is checked before tool length/tool radius compensation after ATC operation. 0: Invalid 1: Valid	
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F158 (bit 1)	—	Invalid	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
F159 (bit 0)	Timing of data creation for the MACHINING NAVIGATION - RESULT display with workpiece counting made valid	Set the timing of data creation for the MACHINING NAVIGATION - RESULT display when "1" is specified for both CONTI. and NUMBER items of the end unit (program repetition with workpiece counting made valid). 0: At the end of the first piece machining 1: At the end of the last piece machining	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F159 (bit 1)	Modal selection at power on or at reset (G64/G61.1)	Modal selection at power on or at reset 0: G64 1: G61.1	
	Program type	E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F159 (bit 2)	—	Fixed value	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F159 (bit 3)	 — Program type — Conditions — Unit — Setting range —		Fixed value
F159 (bit 4)	Whether or not to display on the POSITION display a spindle output diagram stored in the setting file		<p>When an output diagram is displayed for continuous rating in the spindle load meter area of the POSITION display, a spindle output diagram registered in the setting file is:</p> <p>0: Not displayed (An output diagram is displayed using the values set with the parameters SA67 to SA80.)</p> <p>1: Displayed (An output diagram is displayed using the values stored in the setting file.)</p> <p>Note : This parameter is available only to models that allow the operator to switch between load meter ratings and are capable of displaying a spindle output diagram on the POSITION display.</p>
F160 (bit 7)	Priority change when erasing the tool path from the TOOL PATH CHECK display or TRACE display		<p>Priority when erasing the tool path from the TOOL PATH CHECK display or TRACE display</p> <p>0: Drawing first After the tool path is erased, the geometry will be drawn correctly. However, the tool path erasing time will be longer than that in the case when the setting value is "1".</p> <p>1: Erasing time first The tool path erasing time will be shorter than that in the case when the setting value is "0". However, after the tool path is erased, the color of overlapping lines may change from the one before erasure.</p> <p>* The drawing color differs depending on the machine.</p>

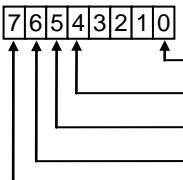
Classification	USER	Display title	EIA/ISO								
Address	Meaning	Description									
F161	—	<p>Lathing tool offset functions</p> <p>F161 bit 0: TOO□□ command: <ul style="list-style-type: none"> For offset number separation OO is used for shape offset. □□ is used for wear offset. For offset number non-separation Offset number □□ is used for both shape offset and wear offset. </p>									
	<table border="1"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>Binary, eight digits</td> </tr> </table>	Program type	M	Conditions	Immediate	Unit	Bit	Setting range	Binary, eight digits	<p>During independent start of tool tip point control</p> <p>0: Movement according to the particular amount of offset 1: No movement according to the particular amount of offset</p>	
Program type	M										
Conditions	Immediate										
Unit	Bit										
Setting range	Binary, eight digits										
F162 (bit 0)	Movement/no movement according to the particular amount of offset during independent start of tool tip point control	<table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>After stop of movement</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>0, 1</td> </tr> </table>		Program type	M, E	Conditions	After stop of movement	Unit	Bit	Setting range	0, 1
Program type	M, E										
Conditions	After stop of movement										
Unit	Bit										
Setting range	0, 1										

Classification	USER	Display title	EIA/ISO								
Address		Meaning									
F162 (bit 1)		<p>Type of passage of tool tip point through singular point</p>  <p>0: The same primary rotary axis angle sign as that existing during the start of tool tip point control is selected as the angle of the rotary axis for implementing the tool axial vector specified during tool tip point control.</p> <p>Example :</p> <p>B = positive B = 0 B = positive</p> <p>C = 0 B = 0 C = 180</p>									
<table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>After stop of movement</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>0, 1</td> </tr> </table>		Program type	M, E	Conditions	After stop of movement	Unit	Bit	Setting range	0, 1	<p>1: The angle that provides a smaller amount of rotational movement of the secondary rotary axis on a singular point is selected as the angle of the rotary axis for implementing the tool axial vector specified during tool tip point control. (Both positive and negative signs are available for the primary rotary axis.)</p> <p>Example :</p> <p>B = positive B = 0 B = negative</p> <p>C = 0 B = 0 C = 0</p>	
Program type	M, E										
Conditions	After stop of movement										
Unit	Bit										
Setting range	0, 1										
F162 (bit 2)		<p>Chamfer/corner R-command address selection</p> <p>0 : (I), (K), (R), (,R), (,C), (,A) 1 : (R), (C), (A)</p>									
<table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>After stop of movement</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>0, 1</td> </tr> </table>		Program type	M, E	Conditions	After stop of movement	Unit	Bit	Setting range	0, 1		
Program type	M, E										
Conditions	After stop of movement										
Unit	Bit										
Setting range	0, 1										
F162 (bit 3)		<p>Fixed hole-machining cycle return selection</p> <p>0: Initial point 1: R-point</p>									
<table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>After stop of movement</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>0, 1</td> </tr> </table>		Program type	M, E	Conditions	After stop of movement	Unit	Bit	Setting range	0, 1		
Program type	M, E										
Conditions	After stop of movement										
Unit	Bit										
Setting range	0, 1										

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F162 (bit 4)	6 digits in T-command for turning		6 digits in T-command for turning 0: Invalid 1: Valid
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
F162 (bit 5)	Use of the M Pro scheme as the method of selecting the Length correction axis bit		Use of the M Pro scheme as the method of selecting the Length correction axis bit 0: Invalid 1: Valid
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
F162 (bit 6)	MAZATROL program check for missing Z-offset Z-offset, valid/invalid		MAZATROL program check for missing Z-offset 0: Valid 1: Invalid
	Program type	M	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
F162 (bit 7)	Encoder polarity selection Encoder polarity selection		Encoder polarity selection 0: Pulse rate increase for forward rotation command 1: Pulse rate reduction for forward rotation command
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	

Classification	USER		Display title	EIA/ISO
Address	Meaning		Description	
F163 (bit 0)	Bar feeder scheduling function		0: Invalid 1: Valid	
	Program type	M, E		
	Conditions	After stop of movement		
	Unit	Bit		
	Setting range	0, 1		
F163 (bit 1)	Incorporation of wear offset data into the current position display in EIA/ISO program mode, valid/invalid		0: Invalid 1: Valid	
			*1 This parameter is valid only for the models for which the G code type for machining centers (F30 = 0) is set and those equipped with the swivel type head.	
			*2 This parameter is valid in automatic operation and virtual machining, but not valid in tool path check.	
	Program type	E		
	Conditions	After stop of movement		
F163 (bit 2)	Incorporation of wear offset data into the current position display in MAZATROL program mode, valid/invalid		0: Invalid 1: Valid	
	Program type	M		
	Conditions	After stop of movement		
	Unit	Bit		
F163 (bit 3)	Position of thread turning tool nose on the VIRTUAL MACHINING display		Position of thread turning tool nose on the VIRTUAL MACHINING display	
	0: Tool tip 1: Position measured using TOOL EYE			
	Program type	M, E		
	Conditions	After stop of movement		
	Unit	Bit		
Setting range		0, 1		

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F163 (bit 4)	Barrier check on the VIRTUAL MACHINING display valid/invalid	0: Invalid 1: Valid	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
F163 (bit 5)	Menu on the DATA I/O display (floppy disk), displayed/hidden	0 : Hidden 1 : Displayed	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
F163 (bit 6)	Menu on the DATA I/O display (tape), displayed/hidden	0 : Hidden 1 : Displayed	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
F163 (bit 7)	Menu on the DATA I/O display (CMT), displayed/hidden	0 : Hidden 1 : Displayed	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F164	Automatic tool data setting conditions		 <ul style="list-style-type: none"> Search for magazine (turret) tool No search for invalid tool No search for broken tool No search for tool that has expired in life No search for tool for which life warning has been issued
	Program type	M	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	

Note :

This parameter is used for automatic setting of the following tool data during MAZATROL program editing:

- “Nominal diameter” and “Suffix” in tool sequence of turning tool
- “Suffix” in tool sequence of point machining

Classification	USER	Display title	EIA/ISO								
Address	Meaning	Description									
F165	—										
		<p>High-speed synchronous tapping function 0: Valid 1: Invalid</p>									
		<p>X-axis movement to minus side during polar coordinate interpolation 0: Enabled 1: Disabled</p>									
		<p>Polar coordinate interpolation of the C-axis 0: Specification in terms of radius value 1: Specification in terms of diameter value</p>									
		<p>C-axis indexing when EIA subprogram is called from MAZATROL program 0: Not executed 1: Executed</p>									
		<p>Modal or non-modal state of Q command in deep hole drilling cycle 0: Modal 1: Non-modal</p>									
		<p>Conversion of tool set data for milling tool based on head swivel angle when G53.5 is commanded 0: Disabled 1: Enabled</p>									
		<p>Behavior of automatic operation of an EIA program when Z-offset is not set 0: Z-offset = 0 1: Alarm stop</p>									
		<p>Setting at CONTI. of the END unit during tool path check 0: Invalid 1: Valid</p>									
<p>Supplemental explanation of F165 Bit 4</p> <p>This bit is used to select whether the depth of infeed per pass (address Q command) in deep hole drilling cycles (G83, G283, and G287 for G-code series M; G83 and G87 for G-code series A) is modal or non-modal.</p> <p>0: Modal The modal value is used if the Q command is not specified. If the modal value of the Q command is 0 (typically, in cases where the Q setting is omitted in the initial deep hole drilling cycle command), or if the Q value is set to 0, an alarm 809 ILLEGAL NUMBER INPUT occurs.</p> <p>1: Non-modal If the Q setting is omitted, or if the Q value is set to 0, the workpiece is cut to the preset infeed depth in a single pass.</p>											
<table border="1"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>After stop of movement</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>Binary, eight digits</td> </tr> </table>		Program type	M	Conditions	After stop of movement	Unit	Bit	Setting range	Binary, eight digits		
Program type	M										
Conditions	After stop of movement										
Unit	Bit										
Setting range	Binary, eight digits										

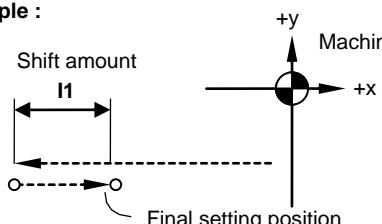
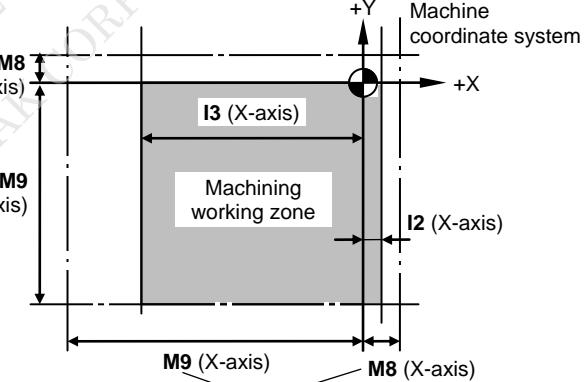
Classification	USER	Display title	EIA/ISO								
F166	<p>—</p> <table border="1"> <tr> <td>Program type</td><td>M, E</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>Bit</td></tr> <tr> <td>Setting range</td><td>Binary, eight digits</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	Bit	Setting range	Binary, eight digits		<p>Description</p> <p>Alteration of tool set value (tool length) on the TOOL DATA display in the automatic operation mode 0: Enabled 1: Disabled</p> <p>Type of wear offset indicated in the milling tool list on the TOOL DATA display 0: Cutting edge offset 1: Wear offset</p> <p>ID No./Tool name selection on the TOOL DATA display 0: ID No. 1: Tool Name</p> <p>Select whether the to-be-drawn plane that has been selected during plane selection on the TRACE or TOOL PATH CHECK display is to be held as the same plane between the two displays or independently between them. 0: Hold as the same plane 1: Independent hold</p> <p>Operation conducted when the [RESTART] menu key is pressed following completion of the search in EIA modal restarting mode 0: Retains the completed status of the search 1: Cancels search results</p> <p>Tool diameter auto-setting for the end mill, face mill, and ball end mill on the TOOL DATA display (these tool diameters are set up during suffix setting automatically.) 0: Automatic setting to nominal diameter 1: No automatic setting</p> <p>Display of the “INTELLIGENT SAFETY SHIELD status specification menu” in the manual operation mode 0: Displayed by the menu selection key 1: Displayed by the [SAFETY SHIELD CANCEL] menu function</p> <p>Automatic backup function 0: Invalid 1: Valid</p>
Program type	M, E										
Conditions	Immediate										
Unit	Bit										
Setting range	Binary, eight digits										

Classification	USER	Display title	EIA/ISO								
Address	Meaning	Description									
F167	—	<p>Threading (by turning) shift angle (Q-command) unit 0: 1/1000° 1: 1°</p> <p>Handling the diameter/radius of point R during the hole-machining fixed cycle 0: Based on the settings for the X-axis 1: Fixed at the radius value</p> <p>Handling the tool measured using the G137H command 0: Regarding the tool as broken 1: Not regarding the tool as broken</p> <p>Handling the single-process mode when operation is completed/reset 0: Not holding the mode 1: Holding the mode</p> <p>If the arc internal correction value is greater than the arc radius 0: Connecting the path linearly 1: Regarding the status as an alarm</p> <p>Handling the X-axis command in G68.5 0: Radius value 1: Diameter value</p> <p>Wear correction direction in the lathe-based machine model 0: Actual axis direction 1: Virtual axis direction</p> <p>Updating the current position by a press of the reset key in manual operation mode 0: Valid 1: Invalid</p>									
<table border="1"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>After stop of movement</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>Binary, eight digits</td> </tr> </table>		Program type	M	Conditions	After stop of movement	Unit	Bit	Setting range	Binary, eight digits	<p>Axis movement at the time when dynamic offset II is canceled (G54.2P0) 0: Perform 1: Do not perform</p>	
Program type	M										
Conditions	After stop of movement										
Unit	Bit										
Setting range	Binary, eight digits										
F168 (bit 0)	Selection of operation to perform when dynamic offset II is canceled										
<table border="1"> <tr> <td>Program type</td> <td>E</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>0, 1</td> </tr> </table>		Program type	E	Conditions	Immediate	Unit	Bit	Setting range	0, 1		
Program type	E										
Conditions	Immediate										
Unit	Bit										
Setting range	0, 1										
F168 (bit 2)	Rotational center offset is added/not added to the tool length when the tool axial tool length offset command (G43.1) is issued	<p>Select whether the rotational center offset is to be added to the tool length when the tool axial tool length offset command (G43.1) is issued</p> <p>F168 (bit 2) = 0: Offset is added.</p> <p>F168 (bit 2) = 1: Offset is not added.</p>									
<table border="1"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>—</td> </tr> <tr> <td>Setting range</td> <td>0, 1</td> </tr> </table>		Program type	M	Conditions	Immediate	Unit	—	Setting range	0, 1		
Program type	M										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										

Classification	USER	Display title	EIA/ISO
Address	Meaning		Description
F168 (bit 3)	Axis for the milling tool wear correction and TOOL EYE compensation amount for the B-axis rotational turret		Select axis for the milling tool wear correction and TOOL EYE compensation amount for the B-axis rotational turret. F168 (bit 3) = 0: Virtual axis F168 (bit 3) = 1: Actual axis
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F168 (bit 4)	Conversion of 3D coordinate conversion into inclined-surface machining		Convert the 3D coordinate conversion (G68/G68.5) into inclined-surface machining (G68.2 or G68.4). 0: Invalid 1: Valid * When G68 is replaced with G68.2 (or G68.4) by setting bit 4 of parameter F168 to 1, 5-axis machining function (inclined-surface machining, workpiece placement error correction) can be used at the same time.
	Program type	E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
F168 (bit 5)	Holding the amount of manual handle pulse interruption		Select whether to hold the amount of shift in spite of pressing the RESET key when manual handle pulse interruption is valid. F168 (bit 5) = 0: Hold F168 (bit 5) = 1: No hold (M640T compatible mode) The same operation as that of the M640T can be selected. This parameter is valid when F95 bit 6 = 0 (holds the amount of manual handle pulse interruption, in spite of the RESET key operation/M30/M02).
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
F168 (bit 6)	[RESTART2NONMODAL] menu function during stop of an EIA program execution invalid/valid		Select whether the [RESTART2NONMODAL] menu function during stop of an EIA program execution is to be made valid or invalid. F168 (bit 6) = 0: Invalid F168 (bit 6) = 1: Valid
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	EIA/ISO
Address	Meaning	Description	
F168 (bit 7)	Correction pattern for groove/ protrusion width measurement		Whether to allow for groove/protrusion width measurement the selection of a correction pattern which assumes that both ends of the groove or protrusion are machined with the same tool
			F168 (bit 7) = 0: Impossible to select the correction pattern
			F168 (bit 7) = 1: Possible to select the correction pattern
	Program type	M	
	Conditions	Immediate	
Unit	—		
	Setting range	0, 1	

2-3-5 User parameter SOFT LIMIT (I)

Classification	USER	Display title	SOFT LIMIT
Address	Meaning		Description
I1	Shift amount of unidirectional positioning (G60)		<p>The amount and direction of shift from the final setting position during unidirectional positioning of the point-machining or during execution of G60.</p> <p>I1 < 0: Positioning in minus direction I1 > 0: Positioning in plus direction</p> <p>Example :</p>  <p>Machine coordinate system +y +x Shift amount I1 Final setting position</p> <p>Note : For the axes which operate in submicrons in the case of submicron machine specifications, the setting unit of this parameter is reduced to 1/10 times.</p> <p>MPL091</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 in (0.0001°)	
	Setting range	0 to ±99999999	
I2	Upper (plus direction) user soft-limit		<p>The parameter used to define the machine working zone in order to prevent machine interference with the workpiece or jigs. Set the coordinate values of the machine coordinate system.</p> <p>Example :</p>  <p>Machine coordinate system +Y +X M8 (Y-axis) I3 (X-axis) M9 (Y-axis) I2 (X-axis) M9 (X-axis) M8 (X-axis) Manufacturer soft-limit</p> <p>MPL092</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 in (0.0001°)	
	Setting range	0 to ±99999999	
I3	Lower (minus direction) user soft-limit		<p>If the machine is likely to overstep its working zone, an alarm will occur and the machine will stop.</p> <p>Note :</p> <ol style="list-style-type: none"> These parameters are valid only when bit 2 of I14 is 0. These parameters are invalid if I2 = I3. For the axes which operate in submicrons in the case of submicron machine specifications, the setting unit of this parameter is reduced to 1/10 times.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 in (0.0001°)	
	Setting range	0 to ±99999999	

Classification	USER	Display title	SOFT LIMIT							
I5	<p>Function for making the G0 speed variable Variable override: Minimum value</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>%</td></tr> <tr><td>Setting range</td><td>0 to 100</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	%	Setting range	0 to 100	<p>Override (%)</p> <p>Axis position</p>
Program type	M, E									
Conditions	Immediate									
Unit	%									
Setting range	0 to 100									
I7	<p>Function for making the G0 speed variable Variable control area (lower limit side)</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001 in (0.0001°)</td></tr> <tr><td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.0001 mm/0.00001 in (0.0001°)	Setting range	0 to 99999999	<p>Note :</p> <ol style="list-style-type: none"> 1. When I7 = 0, the G0 speed variable control function is invalid. 2. When I8 = 0, the variable control area width (upper limit side and lower limit side) is determined by parameter I7. <p>Override (%)</p> <p>Axis position</p>
Program type	M, E									
Conditions	Immediate									
Unit	0.0001 mm/0.00001 in (0.0001°)									
Setting range	0 to 99999999									
I8	<p>Function for making the G0 speed variable, Variable control area (upper limit side)</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001 in (0.0001°)</td></tr> <tr><td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.0001 mm/0.00001 in (0.0001°)	Setting range	0 to 99999999	<p>Note :</p> <ol style="list-style-type: none"> 1. When I7 = 0, the G0 speed variable control function is invalid. 2. When I8 = 0, the variable control area width (upper limit side and lower limit side) is determined by parameter I7. <p>Override (%)</p> <p>Axis position</p>
Program type	M, E									
Conditions	Immediate									
Unit	0.0001 mm/0.00001 in (0.0001°)									
Setting range	0 to 99999999									

Classification	USER	Display title	SOFT LIMIT
Address	Meaning	Description	
I9	Function for making the G0 speed variable Variable control area lower limit	Override (%) Program type M, E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to 99999999	
I10	Function for making the G0 speed variable Variable control area upper limit	Override (%) Program type M, E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to 99999999	
I11	Rotary center of a workpiece	Set the rotary center of a workpiece at a table angle of 0° for each axis in the machine coordinate system. (Valid only with dynamic offset function and in manual operation) Note : For the axes which operate in submicrons in the case of submicron machine specifications, the setting unit of this parameter is reduced to 1/10 times.	
I12	Clamping value for the amount of handle interruption	Set the clamping value for the amount of handle interruption. Program type M, E Conditions After stop of movement Unit 0.001 mm/0.0001 in Setting range 0 to 99999999	

Classification	USER	Display title	SOFT LIMIT
Address	Meaning	Description	
I13	—	<p>Execution of G28 (reference-point return): 0: Memory-type zero-point return 1: Watchdog-type zero-point return</p> <p>Manual zero-point return operation: 0: Memory-type zero-point return (After power-on, however, watchdog-type zero-point return) 1: Watchdog-type zero-point return</p> <p>Removal of control axes 0: No (Not removed) 1: Yes (Removed)</p>	
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
I14	—	<p>Mirror image with respect to the machine zero-point 0: Invalid 1: Valid</p> <p>User software limits (I2, I3) 0: Valid 1: Invalid</p> <p>Tool-tip relief after spindle orientation during execution of G75, G76, G86 or point-machining (boring or back-boring) 0: Required 1: Not required</p> <p>Direction of the relief mentioned above 0: Plus 1: Minus</p>	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
I17	Amount of work hand rotational error correction (Work hand angle: 0°)	<p>Enter the error between the zero point of the actual workpiece and the FRM zero point that has been automatically calculated as a result of the workpiece transfer unit (with work hand angle of 0°).</p> <p>Automatically calculated workpiece position</p> <p>Actual workpiece position</p> <p>I17 (X)</p> <p>I17 (Y)</p> <p>I17 (Z)</p> <p>(Only for machines with a workpiece handling device)</p>	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 in	
	Setting range	-32768 to 32767	

Classification	USER	Display title	SOFT LIMIT
Address		Meaning	
I18		Amount of work hand rotational error correction (Work hand angle: 45°)	
		Program type	M, E
		Conditions	After stop of movement
		Unit	0.0001 mm/0.00001 in
		Setting range	-32768 to 32767
I19		Amount of work hand rotational error correction (Work hand angle: 90°)	
		Program type	M, E
		Conditions	After stop of movement
		Unit	0.0001 mm/0.00001 in
		Setting range	-32768 to 32767
I21	Fixed point 1 specified by the user		Specify the position of the fixed point which is used when 9 is specified in parameter SU10 or that [USER 1 (I21)] is specified in "Change position" of the manual program unit.
I22		Fixed point 2 specified by the user	
		Program type	M
		Conditions	Immediate
		Unit	0.0001 mm/0.00001 in
		Setting range	-99999999 to 99999999

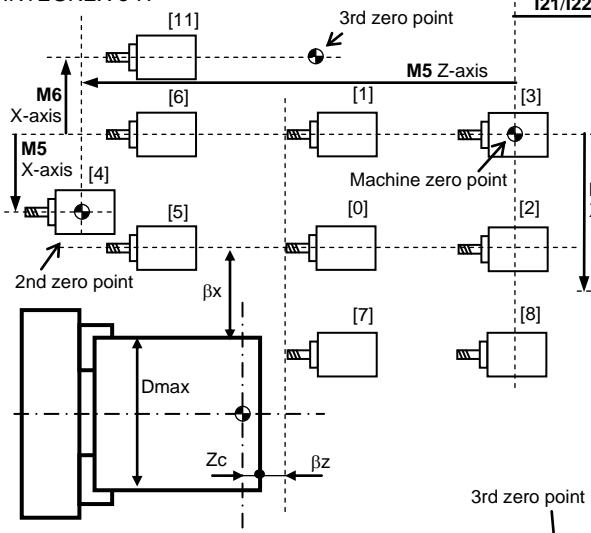
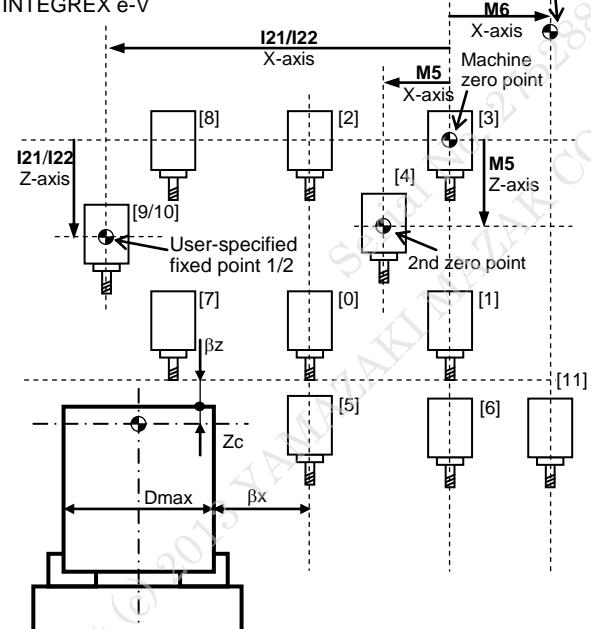
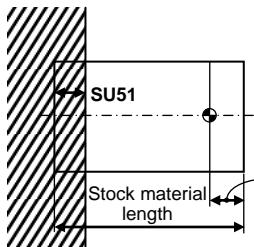
2-3-6 User parameter SYSTEM (SU)

Classification	USER	Display title	SYSTEM
Address	Meaning		Description
SU1	Reference axis of abscissa for plane selection		Set the reference axis of abscissa.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 255	
SU2	Axis 1 parallel to the axis of abscissa for plane selection		Set axis 1 parallel to the axis of abscissa.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 255	
SU3	Axis 2 parallel to the axis of abscissa for plane selection		Set axis 2 parallel to the axis of abscissa.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 255	
SU4	Reference axis of ordinate for plane selection		Set the reference axis of ordinate.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 255	

Classification	USER	Display title	SYSTEM
Address	Meaning		Description
SU5	Axis 1 parallel to the axis of ordinate for plane selection		Set axis 1 parallel to the axis of ordinate.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 255	
SU6	Axis 1 parallel to the axis of ordinate for plane selection		Set axis 2 parallel to the axis of ordinate (cylindrical interpolation).
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 255	
SU7	Reference height axis for plane selection		Set the reference height axis.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 255	
SU8	Axis 1 parallel to the height axis for plane selection		Set axis 1 parallel to the height axis.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 255	

Classification	USER	Display title	SYSTEM
Address	Meaning		Description
SU9	Axis 2 parallel to the height axis for plane selection		Set axis 2 parallel to the height axis.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 255	

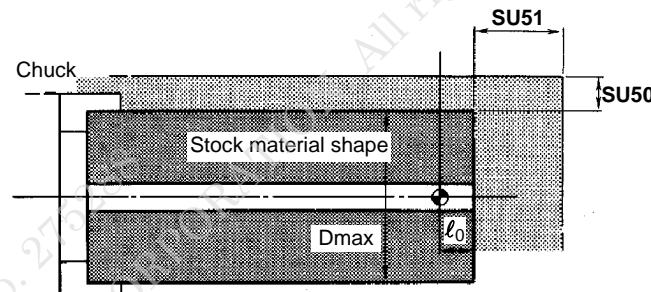
Classification	USER	Display title	SYSTEM																																				
Address	Meaning	Description																																					
SU10	Selection of tool change command output position position	<p>Specify tool change command output position from [0] through [10] below.</p> <table border="1"> <thead> <tr> <th>Setting</th><th>X-axis</th><th>Z-axis</th></tr> </thead> <tbody> <tr> <td>0</td><td>Clearance position</td><td>Clearance position</td></tr> <tr> <td>1</td><td>Machine zero point</td><td>Clearance position</td></tr> <tr> <td>2</td><td>Clearance position</td><td>Machine zero point</td></tr> <tr> <td>3</td><td>Machine zero point</td><td>Machine zero point</td></tr> <tr> <td>4</td><td>Fixed point</td><td>Fixed point</td></tr> <tr> <td>5</td><td>Clearance position</td><td>End point of previous machining</td></tr> <tr> <td>6</td><td>Machine zero point</td><td>End point of previous machining</td></tr> <tr> <td>7</td><td>End point of previous machining</td><td>Clearance position</td></tr> <tr> <td>8</td><td>End point of previous machining</td><td>Machine zero point</td></tr> <tr> <td>9</td><td>User-specified fixed point 1</td><td>User-specified fixed point 1</td></tr> <tr> <td>10</td><td>User-specified fixed point 2</td><td>User-specified fixed point 2</td></tr> </tbody> </table>		Setting	X-axis	Z-axis	0	Clearance position	Clearance position	1	Machine zero point	Clearance position	2	Clearance position	Machine zero point	3	Machine zero point	Machine zero point	4	Fixed point	Fixed point	5	Clearance position	End point of previous machining	6	Machine zero point	End point of previous machining	7	End point of previous machining	Clearance position	8	End point of previous machining	Machine zero point	9	User-specified fixed point 1	User-specified fixed point 1	10	User-specified fixed point 2	User-specified fixed point 2
Setting	X-axis	Z-axis																																					
0	Clearance position	Clearance position																																					
1	Machine zero point	Clearance position																																					
2	Clearance position	Machine zero point																																					
3	Machine zero point	Machine zero point																																					
4	Fixed point	Fixed point																																					
5	Clearance position	End point of previous machining																																					
6	Machine zero point	End point of previous machining																																					
7	End point of previous machining	Clearance position																																					
8	End point of previous machining	Machine zero point																																					
9	User-specified fixed point 1	User-specified fixed point 1																																					
10	User-specified fixed point 2	User-specified fixed point 2																																					
<p>Note :</p> <ol style="list-style-type: none"> SU10 = 5 or 6, Z-axis tool change command output position is identical with the end point of previous machining. In the case below, however, this may not be applied. As shown here, if the longest tool comes into the hatched portion, the position will escape in Z-axis direction by the distance determined by SU51. 																																							
<table border="1"> <tr> <td>Program type</td><td>M</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>—</td></tr> <tr> <td>Setting range</td><td>0 to 10</td></tr> </table>		Program type	M	Conditions	Immediate	Unit	—	Setting range	0 to 10																														
Program type	M																																						
Conditions	Immediate																																						
Unit	—																																						
Setting range	0 to 10																																						

Classification	USER	Display title	SYSTEM																																					
Address	Meaning	Description																																						
SU10	Selection of tool change command output position position	Specify tool change command output position from [0] through [11] below.																																						
	INTEGREX e-H	 <p>Diagram illustrating the INTEGREX e-H tool change mechanism. It shows a sequence of tool changes at different positions along the X-axis and Z-axis. The diagram includes labels for M6, M5, and I21/I22 axes, as well as points like Machine zero point, 2nd zero point, 3rd zero point, and User-specified fixed point 1/2. Dimensions shown include Dmax (stock material maximum outside diameter), Zc (stock material edge projection length), and clearance values β_x and β_z.</p>																																						
	INTEGREX e-V	 <p>Diagram illustrating the INTEGREX e-V tool change mechanism. It shows a sequence of tool changes at different positions along the X-axis and Z-axis. The diagram includes labels for M6, M5, and I21/I22 axes, as well as points like Machine zero point, 2nd zero point, and User-specified fixed point 1/2. Dimensions shown include Dmax (stock material maximum outside diameter), Zc (stock material edge projection length), and clearance value β_z.</p>																																						
	<table border="1"> <thead> <tr> <th>Setting</th> <th>X-axis</th> <th>Z-axis</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>Clearance position</td> <td>Clearance position</td> </tr> <tr> <td>1</td> <td>Machine zero point</td> <td>Clearance position</td> </tr> <tr> <td>2</td> <td>Clearance position</td> <td>Machine zero point</td> </tr> <tr> <td>3</td> <td>Machine zero point</td> <td>Machine zero point</td> </tr> <tr> <td>4</td> <td>2nd zero point</td> <td>2nd zero point</td> </tr> <tr> <td>5</td> <td>Clearance position</td> <td>End point of previous machining</td> </tr> <tr> <td>6</td> <td>Machine zero point</td> <td>End point of previous machining</td> </tr> <tr> <td>7</td> <td>End point of previous machining</td> <td>Clearance position</td> </tr> <tr> <td>8</td> <td>End point of previous machining</td> <td>Machine zero point</td> </tr> <tr> <td>9</td> <td>User-specified fixed point 1</td> <td>User-specified fixed point 1</td> </tr> <tr> <td>10</td> <td>User-specified fixed point 2</td> <td>User-specified fixed point 2</td> </tr> <tr> <td>11</td> <td>3rd zero point</td> <td>End point of previous machining</td> </tr> </tbody> </table>		Setting	X-axis	Z-axis	0	Clearance position	Clearance position	1	Machine zero point	Clearance position	2	Clearance position	Machine zero point	3	Machine zero point	Machine zero point	4	2nd zero point	2nd zero point	5	Clearance position	End point of previous machining	6	Machine zero point	End point of previous machining	7	End point of previous machining	Clearance position	8	End point of previous machining	Machine zero point	9	User-specified fixed point 1	User-specified fixed point 1	10	User-specified fixed point 2	User-specified fixed point 2	11	3rd zero point
Setting	X-axis	Z-axis																																						
0	Clearance position	Clearance position																																						
1	Machine zero point	Clearance position																																						
2	Clearance position	Machine zero point																																						
3	Machine zero point	Machine zero point																																						
4	2nd zero point	2nd zero point																																						
5	Clearance position	End point of previous machining																																						
6	Machine zero point	End point of previous machining																																						
7	End point of previous machining	Clearance position																																						
8	End point of previous machining	Machine zero point																																						
9	User-specified fixed point 1	User-specified fixed point 1																																						
10	User-specified fixed point 2	User-specified fixed point 2																																						
11	3rd zero point	End point of previous machining																																						
<p>Note :</p> <ol style="list-style-type: none"> 1. Do not use the setting of SU10 = 4 for the INTEGREX e-II series. A collision with the ATC cover is likely to occur since the ATC position specified in fixed-point M5 is present inside the ATC cover. 2. Since β_x in the case of SU10 = 0, 2, or 5 includes spindle tool length, if the tool is too long (as in the case of a long boring bar), the tool change operation may be software-limited in the +X direction. 3. SU10 = 5 or 6, Z-axis tool change command output position is identical with the end point of previous machining. In the case below, however, this may not be applied. As shown here, if the longest tool comes into the hatched portion, the position will escape in Z-axis direction by the distance determined by SU51. 																																								
 <p>Diagram showing the effect of SU51 on Z-axis tool change position. It illustrates a hatched area representing the stock material edge protrusion length. The diagram shows a tool changing from a longer tool to a shorter one, with the new position being limited by the distance determined by SU51.</p>																																								
Program type	M																																							
Conditions	Immediate																																							
Unit	—																																							
Setting range	0 to 11																																							

Classification	USER	Display title	SYSTEM
Address	Meaning		Description
SU11	Movement of axes during approach		<p>Specify the types of axes to be simultaneously moved when approaching the tool to the next machining area.</p> <p>0: Three axes (X, Y, Z) move at the same time. 1: After Z-axis movement, the X-axis and the Y-axis move at the same time.</p> <p>Note : When PINCHCUT HOLDER or Y TURN. HOLDER is used either of the following movements can be specified.</p> <p>0: After Y-axis movement, the X-axis and the Z-axis move at the same time. 1: The Y-axis, Z-axis and X-axis move in order.</p> <p><Supplementary description></p> <p>This parameter is valid only when "workpiece scheme" is selected in the common unit. If "initial-point scheme" is selected, three axes move simultaneously, irrespective of setting of this parameter.</p>
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
SU12	Rotating position specified in the index unit after tool change		<p>Select the rotating position specified in the index unit after tool change.</p> <p>0: After the tool change, rotation occurs at the position specified in the index unit. For the movement from the completing position of the tool change to the rotating position of the index unit, three axes (X, Y, Z) move at the same time. 1: After the tool change, rotation occurs at the position specified in the index unit. For the movement from the completing position of the tool change to the rotating position of the index unit, the X-axis and the Y-axis move at the same time following completion of Z-axis movement. 2: After the tool change, rotation occurs at the completing position of the tool change.</p>
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1, 2	
SU13	Axis name of the transfer axis		Set the axis name of the transfer axis.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	&0 to &7F	
SU14	Tool nose mark display color on the TOOL PATH CHECK display/ TRACE display		<p>Set the display color for the tool nose marks in each system indicated on the TOOL PATH CHECK and TRACE displays.</p> <p>0: Red (default) 1: Green 2: Blue 3: Yellow 4: Pink 5: Cyan 6: White</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 6	

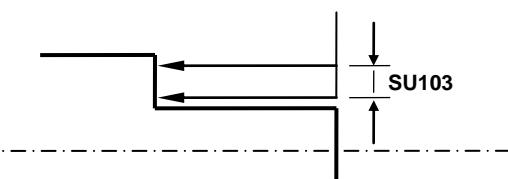
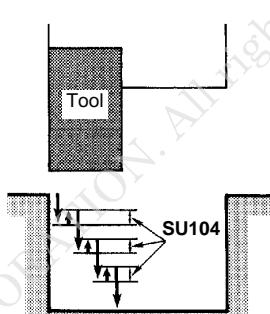
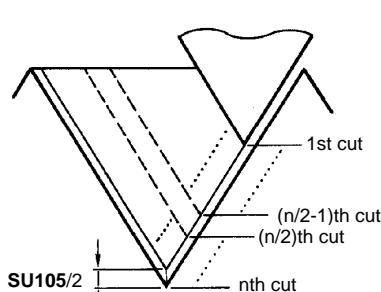
Classification	USER	Display title	SYSTEM
Address	Meaning		Description
SU15	Name of thrust axis for servo axis Program type E Conditions After stop of movement Unit — Setting range 0 to 255		Specify the name of the axis used as the thrust axis for the servo axis.
SU16	Movement to C-axis index swivel position when Z-offset scheme is used Program type M Conditions — Unit — Setting range 0, 1		0: The tool moves to the approach position set in the machining program, without moving to the indexed swivel position. 1: The tool moves to the index swivel position and after C-axis indexing, it moves to the approach position.
SU17	Background color for the MDI window Program type M, E Conditions At power on Unit — Setting range 0 to 8		Set the background color for the MDI window. 0: Green (default) 1: Red 2: Green 3: Blue 4: Yellow 5: Pink 6: Cyan 7: White 8: Black
SU18	Character color for the MDI window Program type M, E Conditions At power on Unit — Setting range 0 to 8		Set the character color for the MDI window. 0: Black (default) 1: Red 2: Green 3: Blue 4: Yellow 5: Pink 6: Cyan 7: White 8: Black

Classification	USER	Display title	SYSTEM
Address	Meaning		Description
SU19	Selecting the method of axis movement to the tool change command output position or to the position designated at RETURN of the end unit		<p>SU19 = 0: Three axes (X, Y, Z) are simultaneously moved to the tool change command output position specified in SU10 and TC62 or to the position designated at RETURN of the end unit.</p> <p>* When the tool nose is present at a position internal to the position of (workpiece diameter + safety profile clearance), the axes pass through the clearance point.</p> <p>SU19 = 1: Following the X-axis, the Y- and Z-axes are simultaneously moved to the tool change command output position specified in SU10 and TC62 or to the position designated at RETURN of the end unit.</p> <p>* When the tool nose is present at a position internal to the position of (workpiece diameter + safety profile clearance), only the X-axis passes through the clearance point. The Z-axis does not pass.</p> <p>Note : This parameter is valid only when the program is of the workpiece scheme (WPC setting) or the Z-offset scheme.</p>
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
SU20	Coefficient for polygonal machining spindle acceleration		<p>During polygonal machining, acceleration of the turning and milling spindle can be decreased by this parameter.</p> <p>Spindle acceleration for polygonal machining = Spindle acceleration × $\frac{SU20}{100}$</p> <p>Note : This value will be handled as 100 if a negative value, 0 or a value more than 100 is set here.</p> <p><Supplementary description> If during polygonal machining alarm 43 SPINDLE MALFUNCTION 2 (52, ,) occurs for the milling spindle, set a value less than 100 in this parameter to elongate the turning spindle speed attainment time (acceleration is decreased).</p>
	Program type	E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
SU21	Selection of returning operation after the execution of the workpiece transfer unit corresponding to the work hand angle to be changed		<p>Specify the axis to be returned home after the execution of the workpiece transfer unit corresponding to the work hand angle to be changed, and the returning sequence.</p> <p>0: X-axis homing → B-axis homing 1: XYZ axes simultaneous homing → B-axis homing 2: X-axis homing → YZ axes simultaneous homing → B-axis homing</p>
	Program type	M	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 2	

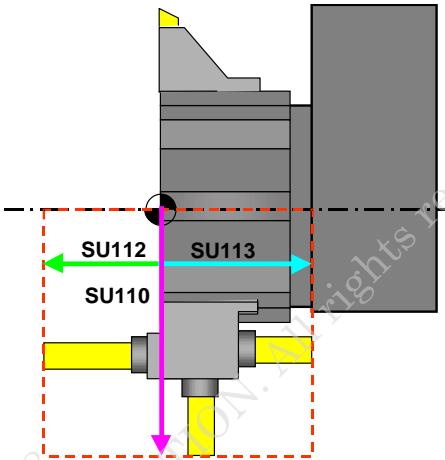
Classification	USER	Display title	SYSTEM
Address	Meaning		Description
SU49	Delay timer for the parts catcher Program type M Conditions Immediate Unit 0.001 s Setting range 0 to 9999		Set dwell time for the parts catcher. Delay timer for the parts catcher
SU50	Tool turning clearance (radial value) in X-axis Program type M Conditions Immediate Unit 0.001 mm/0.0001 in Setting range 0 to 65535		Tool turning clearance is required to prevent interference between the tool and stock material during tool change in automatic operation.  NM211-00220
SU51	Tool turning clearance in Z-axis Program type M Conditions Immediate Unit 0.001 mm/0.0001 in Setting range 0 to 65535		Dmax: Stock material maximum outside diameter l_0 : Stock material edge projection length
SU52 SU53	Lower-turret retraction function Tool number of the retraction tool Program type M Conditions Immediate Unit — Setting range 0 to 960		Set the tool to be retracted to the fixed position for the automatic lower-turret retraction function. Specify the tool number of the lower turret. SU52: Tool number of Retraction tool 1 SU53: Tool number of Retraction tool 2

Classification	USER	Display title	SYSTEM
Address	Meaning	Description	
SU97 to SU100		<p>Set the position where the turret is to be retracted for the turret retraction function. Specify coordinates in the machine coordinate system based on the machine zero point.</p> <p>SU97: X-axis retraction point – Fixed point 1 SU98: Z-axis retraction point – Fixed point 1 SU99: X-axis retraction point – Fixed point 2 SU100: Z-axis retraction point – Fixed point 2</p> <p>Specify a minus value if the fixed point of the retraction position is located in a minus direction when viewed from the zero point of the machine coordinates.</p> <p>Note : Set this parameter for the system of the lower turret.</p>	
Program type	M		
Conditions	Immediate		
Unit	0.001 mm/0.00001 in		
Setting range	0 to 99999999		
SU101	Return distance (radial value) in X-axis at wall during rough cutting in bar machining or in corner machining of EIA/ISO program		<p>Example :</p> <ol style="list-style-type: none"> 1. Standard type cutting Prior to cutting up along the wall in the end of final cycle, escape will be made by specified distance.
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 in	
	Setting range	0 to 65535	
SU102	Return distance in Z-axis at wall during rough cutting in bar machining or in corner machining of EIA/ISO program		<ol style="list-style-type: none"> 2. High speed rough cutting Escape will be made by SU101 and SU102 specified distance during return after reaching the wall.
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 in	
	Setting range	0 to 65535	

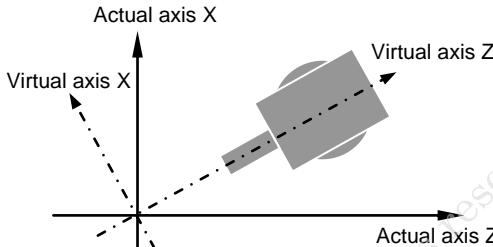
* For MAZATROL programs refer to **TC67** and **TC68**.

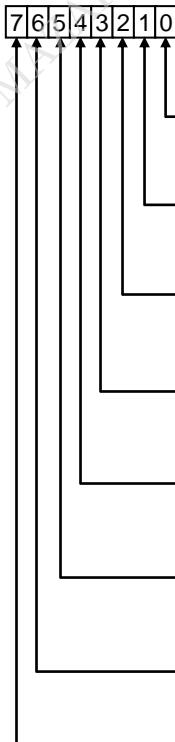
Classification	USER	Display title	SYSTEM
SU103	Cutting depth in the composite-type fixed cycle (G271/G272, G71/G72)		<p>If a cutting depth has not been specified in the program, operation will occur in accordance with the setting of this parameter.</p> 
SU104	Pecking return distance in grooving (G274/G275, G74/G75) of EIA/ISO program	<p>Program type E</p> <p>Conditions Immediate</p> <p>Unit 0.0001 mm/0.00001 in</p> <p>Setting range 0 to 65535</p>	 <p>NM211-00248</p> <p>* For MAZATROL programs refer to TC74.</p>
SU105	Cut depth (diametral value) for final cut in composite-type thread cutting cycle G276, G76 of EIA/ISO program	<p>Program type E</p> <p>Conditions Immediate</p> <p>Unit 0.0001 mm/0.00001 in</p> <p>Setting range 0 to 65535</p>	 <p>NM211-00247</p> <p>* For MAZATROL programs refer to TC78.</p>
SU106	Minimum cut depth clamping value (radial value) in composite-type thread cutting cycle G276 of EIA/ISO program	<p>Program type E</p> <p>Conditions Immediate</p> <p>Unit 0.0001 mm/0.00001 in</p> <p>Setting range 0 to 65535</p>	<p>Clamping will follow the setting of SU106 if the calculated value of the cut depth with the composite type thread cutting cycle G276 is smaller than the setting of SU106. This parameter is valid only for the infeed operation of the fixed volume chip production scheme.</p> <p>* For MAZATROL programs refer to TC79.</p>

Classification	USER		Display title	SYSTEM
Address	Meaning		Description	
SU107	Safety clamp speed		Set the spindle safety clamp speed of a tool whose diameter (nominal diameter) is not specified in the tool data. This clamp speed is invalid for the tools with tool diameter (or nominal diameter, if diameter cannot be specified) setting in the tool data.	
	Program type	M		
	Conditions	Immediate		
	Unit	min ⁻¹		
	Setting range	0 to 99999999		
SU108	Safeguarding strength		Set the machine safeguarding strength to be used during the calculation of the safety speed for the spindle. Remark : Spindle safety speed calculation expression $N = \frac{60 \times 10^3}{\pi D} \sqrt{\frac{2E}{m}} \text{ (min}^{-1}\text{)}$ Tool diameter D (mm): ACT-φ in MAZATROL tool data (For a tool whose tool diameter cannot be registered in tool data: Nominal diameter) Safeguarding strength E (J): Parameter SU108 Tip mass m (kg): Parameter SU109/1000	
	Program type	M		
	Conditions	Immediate		
	Unit	J		
	Setting range	0 to 99999999		
SU109	Tip mass		Set the tip mass used to calculate the spindle safety clamp speed.	
	Program type	M		
	Conditions	Immediate		
	Unit	g		
	Setting range	0 to 99999999		

Classification	USER	Display title	SYSTEM
Address	Meaning	Description	
SU110 to SU113	<p>Tool length setting range check function Settable data range</p> <p>SU110 Max. tool length (+X-axis) SU111 Min. tool length (-X-axis) SU112 Max. tool length (+Z-axis) SU113 Min. tool length (-Z-axis)</p>	<p>Set the desired data range.</p> <p>Note :</p> <p>Although entered data is checked for the range that has been set in SU110 - SU113, the check is conducted only when the "Tool length setting range check function" is valid (SU154 bit 1 = 1).</p> <p>Example of setting (SU111 = 0):</p> 	
SU123	<p>Admissible motion range on the primary rotational axis for workpiece placement error correction</p>	<p>Set the limit for the amount of motion on the primary rotational axis for workpiece placement error correction.</p> <p>Note :</p> <p>Set zero (0) in SU123 if no restriction is to be imposed on the range of correcting motion on the primary rotational axis for workpiece placement error correction.</p>	
SU124	<p>Admissible motion range on the secondary rotational axis for workpiece placement error correction</p>	<p>Set the limit for the amount of motion on the secondary rotational axis for workpiece placement error correction.</p> <p>Note :</p> <p>Set zero (0) in SU124 if no restriction is to be imposed on the range of correcting motion on the secondary rotational axis for workpiece placement error correction.</p>	

Classification	USER	Display title	SYSTEM															
Address	Meaning	Description																
SU125	—	Fixed value																
	Program type	—																
	Conditions	—																
	Unit	—																
	Setting range	—																
SU128 SU129	Retraction positions of the upper and lower turrets at the end of the two-workpiece machining sequence	<p>Set the X- and Z-axis retraction positions of the upper and lower turret in the machine coordinate system. When parameter TC108 = 3, the upper and lower turrets are placed in a queue in this position after the two-workpiece machining sequence.</p>																
	Program type	M																
	Conditions	Immediate																
	Unit	0.0001 mm/0.00001 in																
	Setting range	± 99999999																
SU153 (bit 0)	M-code selection for tapping cycle	<p>M-code selection for tapping cycle.</p> <p>If the command block of G284/G288 (machining G-codes, F30 = 0) or of G84/G88 (turning G-codes, F30 = 1) does not contain spindle forward/reverse rotation M-codes, one of the following M-codes will be output, depending on the status of bit 0 in parameter SU153:</p> <table border="1"> <tr> <td></td><td>BA59 = 3 BA60 = 4</td><td>BA59 = 203 BA60 = 204</td></tr> <tr> <td>SU153</td><td>0</td><td>Tapping cycle of turning</td></tr> <tr> <td>bit 0</td><td>1</td><td>Inverse tapping cycle of turning</td></tr> <tr> <td></td><td></td><td>Tapping cycle of milling</td></tr> <tr> <td></td><td></td><td>Inverse tapping cycle of milling</td></tr> </table> <ul style="list-style-type: none"> If 0 is set in BA59, 3 will be regarded as having been set, and M03 will be output. If 0 is set in BA60, 4 will be regarded as having been set, and M04 will be output. 			BA59 = 3 BA60 = 4	BA59 = 203 BA60 = 204	SU153	0	Tapping cycle of turning	bit 0	1	Inverse tapping cycle of turning			Tapping cycle of milling			Inverse tapping cycle of milling
	BA59 = 3 BA60 = 4	BA59 = 203 BA60 = 204																
SU153	0	Tapping cycle of turning																
bit 0	1	Inverse tapping cycle of turning																
		Tapping cycle of milling																
		Inverse tapping cycle of milling																
	Program type	E																
	Conditions	After stop of movement																
	Unit	—																
	Setting range	0, 1																

Classification	USER	Display title	SYSTEM								
Address	Meaning	Description									
SU153 (bit 1) (bit 2) (bit 3)	<p>Bit 1: Display of BUFFER on the POSITION display</p> <p>Bit 2: Display of REMAIN on the POSITION display</p> <p>Bit 3: Display of POSITION on the POSITION display</p>	<p>Display of BUFFER/REMAIN/POSITION on the POSITION display</p> <p>0: Display in the machine axial direction 1: Display in the axial direction in the coordinates system that has been converted by the inclination of the B-axis</p> 									
	<table border="1"> <tr> <td>Program type</td><td>M, E</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>—</td></tr> <tr> <td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0, 1		
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										
SU153 (bit 4)	<p>Automatic retraction of the lower turret for MAZATROL programs</p>	<p>The retraction of the lower turret in the execution of a MAZATROL program is determined according to the program contents and this parameter's setting as detailed below.</p> <table border="1"> <tr> <td>Program contents</td><td>Retraction of the lower turret</td></tr> <tr> <td>With a retraction command</td><td> <p>Retraction to the position as specified with the [TURRET 2 POS. 1] or [TURRET 2 POS. 2] menu item</p> <p>POS. 1: X = SU97, Z = SU98 POS. 2: X = SU99, Z = SU100</p> </td></tr> <tr> <td>Without retraction commands</td><td> <ul style="list-style-type: none"> SU153 bit 4 = 0: No retraction SU153 bit 4 = 1: Retraction Movement to the position of automatic retraction as below. <p>MULTIPLEX: X/Z = 2nd zero point (M5) Others: X/Y/Z = Machine zero point (M4)</p> </td></tr> </table>	Program contents	Retraction of the lower turret	With a retraction command	<p>Retraction to the position as specified with the [TURRET 2 POS. 1] or [TURRET 2 POS. 2] menu item</p> <p>POS. 1: X = SU97, Z = SU98 POS. 2: X = SU99, Z = SU100</p>	Without retraction commands	<ul style="list-style-type: none"> SU153 bit 4 = 0: No retraction SU153 bit 4 = 1: Retraction Movement to the position of automatic retraction as below. <p>MULTIPLEX: X/Z = 2nd zero point (M5) Others: X/Y/Z = Machine zero point (M4)</p>	<p>Note : Set this parameter for the system of the lower turret.</p>		
Program contents	Retraction of the lower turret										
With a retraction command	<p>Retraction to the position as specified with the [TURRET 2 POS. 1] or [TURRET 2 POS. 2] menu item</p> <p>POS. 1: X = SU97, Z = SU98 POS. 2: X = SU99, Z = SU100</p>										
Without retraction commands	<ul style="list-style-type: none"> SU153 bit 4 = 0: No retraction SU153 bit 4 = 1: Retraction Movement to the position of automatic retraction as below. <p>MULTIPLEX: X/Z = 2nd zero point (M5) Others: X/Y/Z = Machine zero point (M4)</p>										
SU153 (bit 5)	<p>"C-axis Clamp" machining method for MAZATROL programs (Line/Linear machining for the C-axis CYLIND mode)</p>	<p>Set the "C-axis Clamp" machining method for MAZATROL programs (line/linear machining for the C-axis CYLIND mode).</p> <p>0: Machining with C-axis braked 1: Machining with C-axis clamped (only for machining without C-axis movement)</p>									
	<table border="1"> <tr> <td>Program type</td><td>M</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>—</td></tr> <tr> <td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	—	Setting range	0, 1		
Program type	M										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										

Classification	USER	Display title	SYSTEM
Address	Meaning	Description	
SU153 (bit 6)		MAZATROL point machining with C-axis clamped (M210) 0: Output of M210 1: No output of M210	
MAZATROL point machining with C-axis clamped (M210)			
Program type M			
Conditions Immediate			
Unit —			
Setting range 0, 1			
SU153 (bit 7)		Set the "C-axis Clamp" machining method for MAZATROL programs (line/linear machining for the C-axis FACE mode). 0: Machining with C-axis braked 1: Machining with C-axis clamped (only for machining without C-axis movement)	
"C-axis Clamp" machining method for MAZATROL programs (Line/Linear machining for the C-axis FACE mode)			
Program type M			
Conditions Immediate			
Unit —			
Setting range 0, 1			
SU154		 <ul style="list-style-type: none"> 7 6 5 4 3 2 1 0 ↑ ↑ ↑ ↑ ↑ ↑ ↑ ↑ Displaying tool-setting values for ATC-type turret 0: Invalid 1: Valid Tool length setting range check function 0: Invalid 1: Valid Display of BUFFER during G18.2/G18.3 mode 0: Actual axis 1: Virtual axis Display of REMAIN during G18.2/G18.3 mode 0: Actual axis 1: Virtual axis Display of POSITION during G18.2/G18.3 mode 0: Actual axis 1: Virtual axis Display of BUFFER during G17.1 - G17.5 mode 0: Actual axis 1: Virtual axis Display of REMAIN during G17.1 - G17.5 mode 0: Actual axis 1: Virtual axis Display of POSITION during G17.1 - G17.5 mode 0: Actual axis 1: Virtual axis 	
Program type M, E			
Conditions Immediate			
Unit Bit			
Setting range Binary, eight digits			

Classification	USER	Display title	SYSTEM
Address	Meaning		Description
SU155 (bit 0)	 — Program type — Conditions — Unit — Setting range —		Fixed value (1)
SU155 (bit 1)	Correction timing of the touch sensor tool length during EIA/ISO programmed or MDI operation (for machining center specifications only)		Select the timing in which the tool length of the touch sensor is to be made valid during EIA/ISO programmed or MDI operation when the following conditions are satisfied: <ul style="list-style-type: none">• F93 bit 3 = 1 (using MAZATROL tool length)• F94 bit 7 = 0 (using tool offsets) 0: When a move command is executed for any controlled axis (X, Y, Z, or rotational axis) 1: When a move command is executed for the Z-axis
SU155 (bit 2)	Operation at an EIA subprogram call from a MAZATROL program		Select whether the zero point return is to be executed at an EIA subprogram call from a MAZATROL program. 0: Executed 1: Not executed
SU155 (bit 3)	Judgment method for tool change command output during MAZATROL programmed operation		Select whether the index angle of the spindle is to be checked during MAZATROL programmed operation when a judgment on whether a tool change is necessary. 0: No check (No tool change command is output when the tool is the same in tool name, nominal size, and suffix.) 1: Check (No tool change command is output when the tool is the same in tool name, nominal size, suffix, and spindle index angle.)

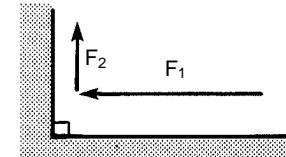
Classification	USER	Display title	SYSTEM
Address	Meaning	Description	
SU155 (bit 4)	Tool change command output at the start of MAZATROL programmed operation		Select whether the tool change command is to be output during MAZATROL programmed operation when the machining unit is executed for the first time. 0: Output of the tool change command except for the same tool 1: Unconditional output of the tool change command
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
SU155 (bit 7)	Reference value for judging tool breakage during laser tool measurement		Select a method of judging tool breakage during laser tool measurement. 0: Judging from the setting of parameter L6 1: Judging from the tolerance specified in the program
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
SU156 (bit 4)	Type of deep hole drilling cycle G83/G87		Type of deep hole drilling cycle G83/G87 0: Normal type 1: High-speed type
	Program type	E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
SU156 (bit 6)	Execution of a restarting search for a block in the mode of G68 when tool length compensation (G43/G44) remains valid with a turning tool mounted.		Execution of the restarting search for a block in the mode of G68 when tool length compensation (G43/G44) remains valid with a turning tool mounted. 0: Alarm 1: Possible (however, the return to the restarting position is not conducted)
	Program type	E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	

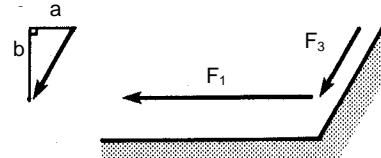
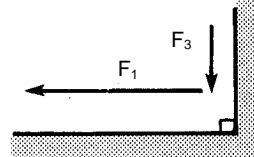
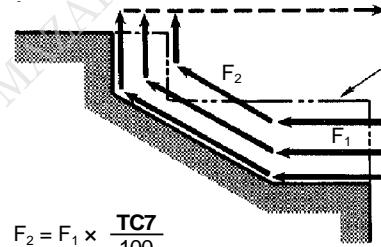
Classification	USER	Display title	SYSTEM
Address	Meaning		Description
SU156 (bit 7)	Execution of a restarting search for a block after cancellation of the cross command	Program type Conditions Unit Setting range	Execution of the restarting search for a block after the cancellation of the cross command 0: Alarm 1: Possible (however, the return to the restarting position is not conducted)
SU157 (bit 1)	The scheme of axis movement during an approach and during movement to the tool change command output position for machining at the work hand side	Program type Conditions Unit Setting range	0: Compliant with SU11 and SU19 . 1: Approach with priority assigned to the Z-axis (SU11 = 1) Movement to the tool rotating position with priority assigned to the X-axis (SU19 = 1) (Only for machines with a workpiece handling device)
SU157 (bit 2)	Conditions for moving to the tool change command output position in manual program unit	Program type Conditions Unit Setting range	Operation pattern with [NO RETURN] selected for CHANGE-PT in the manual program unit (MANL PRG) 0: NO RETURN to the tool change command output position is effective only when the tool need not be changed for another one, nor in the index angle. 1: NO RETURN to the tool change command output position is effective when there is no need of changing the tool, irrespectively of whether its index angle is to be altered or not.
SU157 (bit 3)	Conditions for moving to the tool change command output position for the same TNo.	Program type Conditions Unit Setting range	Response to a tool change command for the same tool number (TNo.) 0: No movement to the tool rotating position 1: Movement to the tool rotating position when either of the following three conditions is satisfied: a. The value of tool length/tool setting data differs. b. The value of nose R/tool diameter differs. c. The section to be machined differs.

Classification	USER		Display title	SYSTEM
Address	Meaning		Description	
SU157 (bit 4)	Operation to perform if either TURN POS X , Y , or Z is left blank in index unit		Select the kind of operation to be performed if either TURN POS X , Y , or Z of the indexing unit is left blank (only when a tool change is not conducted in immediate front of the indexing unit). 0: The unspecified axis moves to the position of the third home position (M6) before the B/C-axes are swiveled. (Conventional operation) 1: The B/C-axes are swiveled without any motion having been conducted on the unspecified axis. (New operation)	
	Program type	M		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
SU157 (bit 5)	Preparatory C-axis positioning to 0° for manual program unit with a milling tool in turning mode		Preparatory C-axis positioning to 0° for a manual program unit (MANL PRG) with a milling tool after a turning unit 0: Positioning to 0° on the C-axis 1: No movement on the C-axis.	
	Program type	M		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
SU157 (bit 7)	Selection of whether or not the B-axis rotation radius is to be included into the "clearance position," a component of the tool change position (when TC62 = 0, 1, 2, 5 or 7) for changing the indexed cutting edge on one and the same FLASH tool.		Select whether or not the B-axis rotation radius is to be included into the "clearance position," a component of the tool change position (when TC62 = 0, 1, 2, 5 or 7) for changing the indexed cutting edge on one and the same FLASH tool. 0: Included 1: Not included	
	Program type	M		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
SU158 (bit 2)	Checking the rotary axis correction values in workpiece placement error correction for MAZATROL programs		0: Invalid 1: Valid Note : Admissible motion ranges are to be specified in SU123 and SU124 respectively for the primary and secondary rotational axis.	
	Program type	M		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		

Classification	USER	Display title	SYSTEM
Address	Meaning	Description	
SU159 (bit 0)	"TOOL NAME ORDER" display/hidden with random ATC machines	0: Displayed 1: Not displayed	
Program type	M, E		
Conditions	Immediate		
Unit	Bit		
Setting range	0, 1		

2-3-7 User parameter TURNING (TC)

Classification	USER	Display title	TURNING
Address	Meaning		Description
TC1	Cut depth reduction rate for rough cutting in bar machining unit, corner machining unit, and copy machining unit		Cut depth can be reduced as remaining workpiece thickness becomes less in rough cutting in bar machining unit, corner machining unit, and copy machining unit. Reduced cut depth (A) can be expressed by $A = T \times \frac{TC1}{100}$ where T = Remaining thickness (radial value).
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
TC2	Acceleration rate in up-going taper for rough cutting in bar machining unit		 <p style="text-align: right;">NM211-00262</p> $F_2 = F_1 \times \frac{TC2}{100} \times \left \frac{b}{a} \right $ <p>F₁ : Feed rate for rough cutting F₂ : Increased feed rate</p>
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	This is effective only when TC141 (bit 0) = 1.
TC3	Acceleration rate in up-going wall slope (90°) for rough cutting in bar machining unit		 <p style="text-align: right;">NM211-00263</p> $F_2 = F_1 \times \frac{TC3}{100}$ <p>F₁ : Feed rate for rough cutting F₂ : Increased feed rate</p>
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	Note : Up to 500 % can be set. This is effective only when TC141 (bit 0) = 1.
TC4	Selection of chamfering pattern in rough external turning cycle G71 (G271) or rough edge-machining cycle G72 (G272)		<p>TC4 = 0: 45° TC4 = 1: Chamfering along the shape to be finished TC4 = 2: Feed rate (F) for chamfering is expressed as follows:</p> $F = F_0 \times \frac{TC3}{100}$ <p>(where F₀ = Feed rate specified in program) When TC4 = 2, feed rate can be changed only when chamfering at 90°.</p> <p>(⇒ TC3)</p>
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

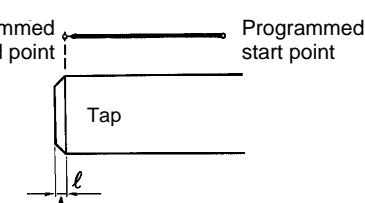
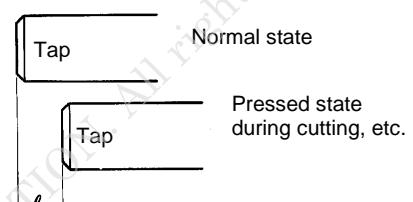
Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC5	Deceleration rate in down-going taper for rough cutting in bar machining unit	 $F_3 = F_1 \times \frac{TC5}{100} \times \left \frac{a}{b} \right $ <p>F₁ : Feed rate for rough cutting F₃ : Reduced feed rate</p> <p>This is effective only when TC141 (bit 0) = 1.</p>	NM211-00264
TC6	Deceleration rate in down-going wall slope (90°) for rough cutting in bar machining unit	 $F_3 = F_1 \times \frac{TC6}{100}$ <p>F₁ : Feed rate for rough cutting F₃ : Reduced feed rate</p> <p>This is effective only when TC141 (bit 1) = 1.</p>	NM211-00265
TC7	Acceleration rate on outside stock contour for rough cutting in copy machining unit	 $F_2 = F_1 \times \frac{TC7}{100}$ <p>F₁ : Feed rate inside stock contour F₂ : Feed rate outside stock contour</p>	NM211-00266
TC8	Acceleration pitch error ratio in thread cutting unit	<p>Used to calculate acceleration distance in thread cutting unit</p> $L = L_0 \left\{ -\ell_n \left(\frac{TC8}{1000} \right) - 1 + \frac{TC8}{1000} \right\}$ <p>L : Acceleration distance L₀ : Distance over which feed rate become constant</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 500	

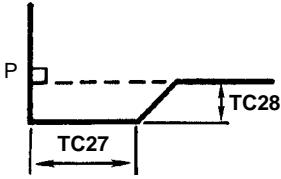
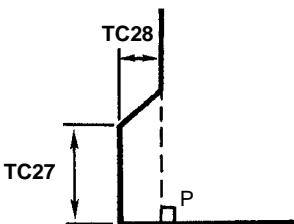
Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC9	Rough cutting residue ratio in cutting off cycle in groove cutting unit	<p> P_S : Programmed start point P_E : Programmed end point l : Groove machining depth $\ell = \overline{P_S \cdot P_E}$ d : Rough cutting residue $d = \ell \times \frac{\text{TC9}}{100}$ </p> <p>NM211-00267</p>	
	Program type M	[1] Cutting at rough cutting feed rate to a point short of end point P_E by distance d	
	Conditions Immediate	[2] Cutting off at finish cutting feed rate to end point P_E	
	Unit %		
	Setting range 0 to 100		
TC10	Cut depth allowable incremental rate for rough cutting in groove cutting unit, edge machining unit and copy machining unit	<p>Used to calculate minimum cutting frequency in groove cutting unit, edge machining unit and copy machining unit</p> $d' = d \left(\frac{100 + \text{TC10}}{100} \right)$ <p>d : Cut depth per cycle d' : Allowable maximum cut depth</p>	
	Program type M		
	Conditions Immediate		
	Unit %		
	Setting range 0 to 100		
TC11	Deceleration rate at cutting start in turning-drilling unit	$F' = F \times \frac{\text{TC11}}{100}$ <p>F : Specified feed rate F' : Feed rate at cutting start</p> <p>End point → Programmed start point</p> <p>Feed rate at F' over distance ℓ from the programmed start point</p>	NM211-00268
	Program type M		
	Conditions Immediate		
	Unit %		
	Setting range 0 to 100	$\begin{cases} \ell = \frac{D}{2 \times \tan\theta/2} & (0^\circ < \theta < 180^\circ) \\ \ell = 0 & (\theta \geq 180^\circ) \end{cases}$	

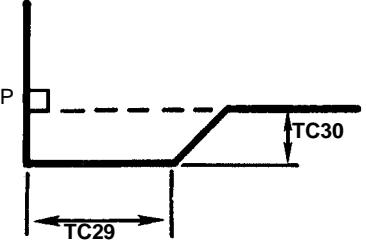
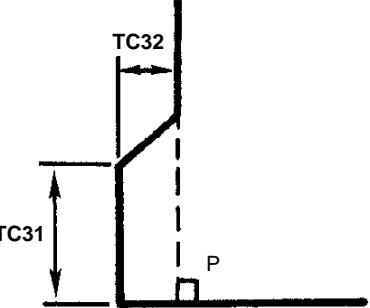
Classification	USER	Display title	TURNING						
Address	Meaning	Description							
TC12	Deceleration rate at cutting end in turning-drilling unit	$F' = F \times \frac{TC12}{100}$ <p>F : Specified feed rate F' : Feed rate at cutting start</p> <p>Programmed end point Programmed start point</p> <p>End point Drill</p> <p>θ</p> <p>D</p> <p>l</p>							
		<p>Feed rate at F' to the point distant by l from the end point</p> <p>NM211-00269</p>							
		<table border="1"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>%</td> </tr> <tr> <td>Setting range</td> <td>0 to 100</td> </tr> </table>		Program type	M	Conditions	Immediate	Unit	%
Program type	M								
Conditions	Immediate								
Unit	%								
Setting range	0 to 100								
$\begin{cases} l = \frac{D}{2 \times \tan\theta/2} + \frac{D}{10} & (0^\circ < \theta < 80^\circ) \\ l = \frac{D}{10} & (\theta \geq 180^\circ) \end{cases}$									
<p>Deceleration clearance (TC15)</p> <p>NM211-00270</p>									
TC13	Deceleration rate at rough cutting start in bar machining unit and copy machining unit	$F' = F \times \frac{TC13}{100}$ <p>F: Specified feed rate F': Feed rate at cutting start</p>							
		<table border="1"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>%</td> </tr> <tr> <td>Setting range</td> <td>0 to 100</td> </tr> </table>		Program type	M	Conditions	Immediate	Unit	%
Program type	M								
Conditions	Immediate								
Unit	%								
Setting range	0 to 100								
<p>(⇒ TC15)</p>									

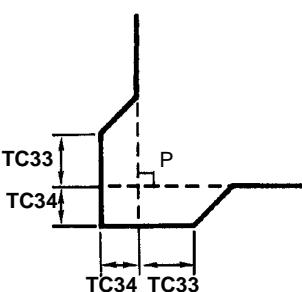
Classification	USER	Display title	TURNING
Address		Meaning	Description
TC14		Maximum permissible rate of increase of the initial cutting depth during roughing	<p>This parameter is valid during initial cutting in a bar roughing cycle. If the remainder obtained by dividing the thickness of cutting during the roughing cycle by the corresponding cutting depth stays within the range specified by this parameter, that value will be added to the initial cutting depth to reduce the cutting repeat times.</p> <p>Example :</p> <p>1. $TC14 = 0$ (R-depth.: 5)</p> <p>2. $TC14 = 10\%$ (R-depth.: 5)</p> <p>Cutting to a depth of 10.1 is executed in 3 rounds of cutting (5, 5, and 0.1). Cutting at a depth setting of 5 results in a final cutting depth of 0.1. Since 10% of the depth setting of 5 is 0.5 and this value is larger than the final cutting depth, this final cutting depth is added to the initial cutting depth.</p>
Program type		M	
Conditions		Immediate	
Unit		%	
Setting range		0 to 100	Note : Up to 100% can be set. When a value larger than 100% is set, it is regarded as 0.
TC15		Deceleration clearance at start of rough cutting in bar machining unit and copy machining unit	<p>Block separated</p> <p>Front clearance (TC39)</p> <p>$F \geq F'$ (Feedrate decelerated for the initial contact with workpiece)</p> <p>TC15</p> <p>NM211-00256</p>
Program type		M	
Conditions		Immediate	
Unit		0.001 mm/0.0001 in	
Setting range		0 to 65535	Note : Using this parameter will reduce load in the initial contact between the tool and workpiece. (⇒TC13)
TC16		Tolerance for escape in high speed rough cutting cycle of bar machining unit	<p>TC16</p> <p>Escape</p> <p>P_1</p> <p>P_2</p> <p>P_3</p> <p>d: Distance in cutting direction between P_1 and P_2</p> <p>NM211-00257</p>
Program type		M	
Conditions		Immediate	
Unit		0.001 mm/0.0001 in	
Setting range		0 to 65535	If $TC16 \geq d$, tool escape from the workpiece. If $TC16 < d$, tool doesn't escape from the workpiece.

Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC17	Pitch error correction during threading acceleration	Set the starting pitch error rate of threading. [1] 	Ideal pitch = Starting pitch of threading ([1]) + TC17
	Program type M Conditions Immediate Unit 0.001 mm Setting range 0 to 40		
TC18	—	Fixed value (0)	
	Program type — Conditions — Unit — Setting range —		
TC19	Turning-drilling cut depth calculation coefficient	Used for automatic determination of first cut depth in turning-drilling $d_1 = D \times \frac{TC19}{100}$ D : Drilling hole diameter d ₁ : Cut depth of first cut	
	Program type M Conditions Immediate Unit % Setting range 0 to 1000		
TC20	Reamer return speed calculation coefficient in the turning-drilling unit	 $F_2 = F_1 \times \frac{TC20}{100}$	NM211-00271 F ₁ : Specified feed rate F ₂ : Return speed P ₁ : Start point P ₂ : End point
	Program type M Conditions Immediate Unit % Setting range 0 to 999		

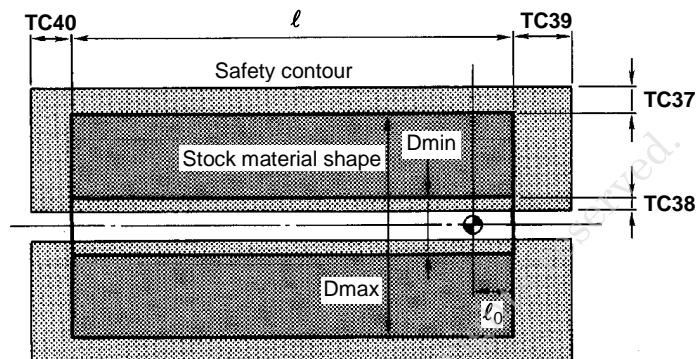
Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC21	Incomplete threading portion length calculation coefficient for turning-tap tip	$\ell = P \times \frac{\text{TC21}}{10}$ P : Tapping pitch ℓ : Incomplete thread portion length	 Programmed end point Programmed start point Tap l Cutting end point specified farther by this length l
	Program type	M	
	Conditions	Immediate	
	Unit	Pitch/10	
	Setting range	0 to 99	NM211-00273
TC22	Turning-tapper elongation calculation coefficient	$\ell = P \times \frac{\text{TC22}}{10}$ P : Tapping pitch ℓ : Tapper elongation	 Normal state Tap l Pressed state during cutting, etc.
	Program type	M	
	Conditions	Immediate	
	Unit	Pitch/10	
	Setting range	0 to 99	NM211-00274
TC23	Thread height calculation coefficient for outside diameter, face/rear thread cutting (metric)	$h = P \times \frac{\text{TC23}}{10000}$ h : Thread height P : Thread pitch	
	Program type	M	
	Conditions	Immediate	
	Unit	0.01%	
	Setting range	0 to 65535	
TC24	Thread height calculation coefficient for inside diameter thread cutting (metric)	$h = P \times \frac{\text{TC24}}{10000}$ h : Thread height P : Thread pitch	
	Program type	M	
	Conditions	Immediate	
	Unit	0.01%	
	Setting range	0 to 65535	

Classification	USER	Display title	TURNING
Address	Meaning		Description
TC25	Thread height calculation coefficient for outside diameter, face/rear thread cutting (inch)		$h = P \times \frac{TC25}{10000}$ <p>h : Thread height P : Thread pitch</p>
	Program type	M	
	Conditions	Immediate	
	Unit	0.01%	
	Setting range	0 to 65535	
TC26	Thread height calculation coefficient for inside diameter thread cutting (inch)		$h = P \times \frac{TC26}{10000}$ <p>h : Thread height P : Thread pitch</p>
	Program type	M	
	Conditions	Immediate	
	Unit	0.01%	
	Setting range	0 to 65535	
TC27	Recessing width for #1 to #3		<p><#1></p>  <p>TC27: Width of the first step down from the surface.</p> <p>TC28: Total depth of the recessing operation.</p> <p>P: Programmed end point.</p> <p>NM211-00277</p>
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 65535	
TC28	Recessing depth #1 to #3		<p><#2></p>  <p>TC27: Width of the first step down from the surface.</p> <p>TC28: Total depth of the recessing operation.</p> <p>P: Programmed end point.</p> <p>NM211-00278</p>
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 65535	

Classification	USER	Display title	TURNING								
Address	Meaning	Description									
TC29	<p>Recessing width for #4</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535	<p><#4></p>  <p>P: Programmed end point</p>	NM211-00280
Program type	M										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										
TC30	<p>Recessing depth for #4</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535		
Program type	M										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										
TC31	<p>Recessing width for #5</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535	<p><#5></p> 	NM211-00278
Program type	M										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										
TC32	<p>Recessing depth for #5</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535	<p>P: Programmed end point</p>	
Program type	M										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										

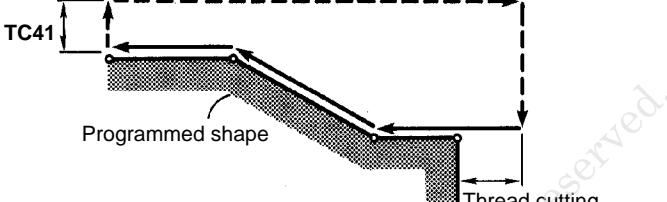
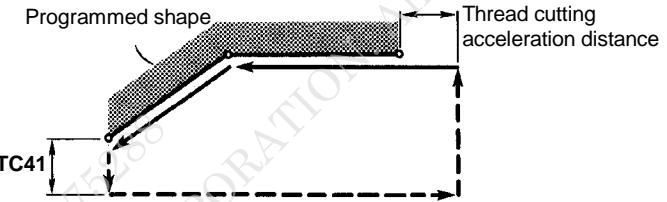
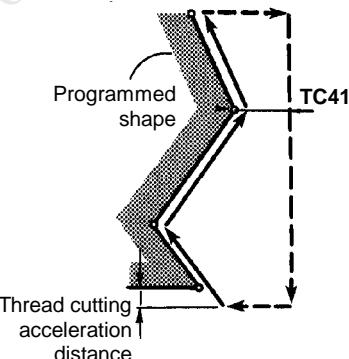
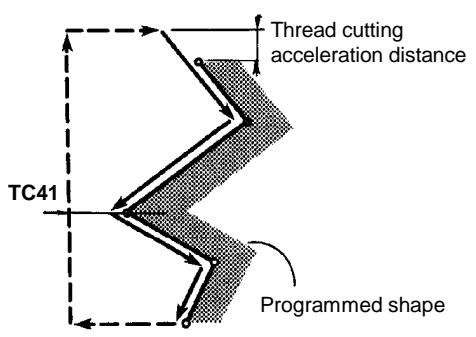
Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC33	Recessing width for #6		<#6>
	Program type M Conditions Immediate Unit 0.001 mm/0.0001 in Setting range 0 to 65535		 P: Programmed end point
	Recessing depth for #6		NM211-00279
	Program type M Conditions Immediate Unit 0.001 mm/0.0001 in Setting range 0 to 65535		
TC34	Recessing depth for #6		
	Program type M Conditions Immediate Unit 0.001 mm/0.0001 in Setting range 0 to 65535		

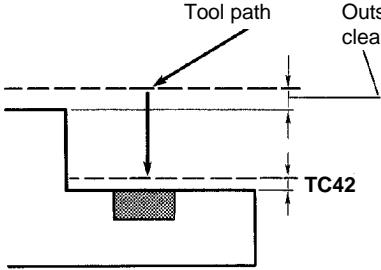
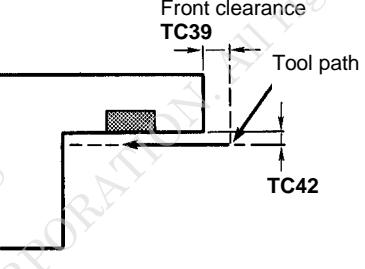
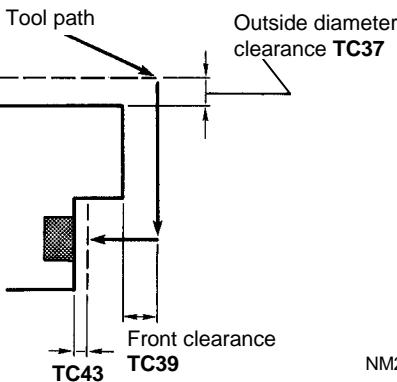
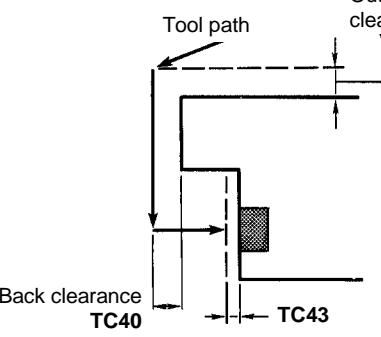
Classification	USER	Display title	TURNING
Address		Meaning	
TC37		Safety contour clearance — Outside diameter clearance (radial value)	
		Program type	M
		Conditions	Immediate
		Unit	0.001 mm/0.0001 in
		Setting range	0 to 65535
TC38		Safety contour clearance — Inside diameter clearance (radial value)	
		Program type	M
		Conditions	Immediate
		Unit	0.001 mm/0.0001 in
		Setting range	0 to 65535
TC39		Safety contour clearance — Front clearance	
		Program type	M
		Conditions	Immediate
		Unit	0.001 mm/0.0001 in
		Setting range	0 to 65535
TC40		Safety contour clearance — Back clearance	
		Program type	M
		Conditions	Immediate
		Unit	0.001 mm/0.0001 in
		Setting range	0 to 65535

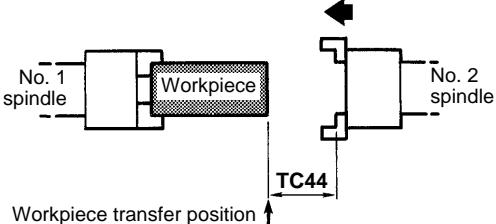
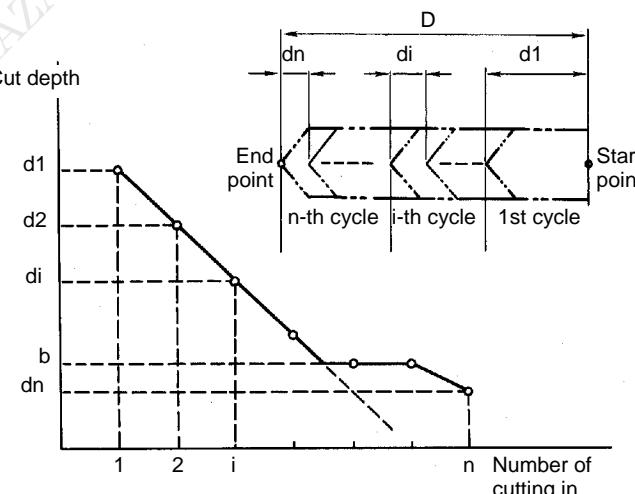


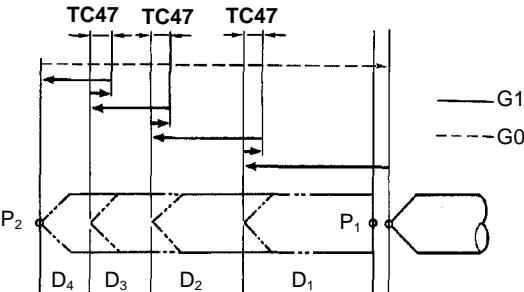
NM211-00221

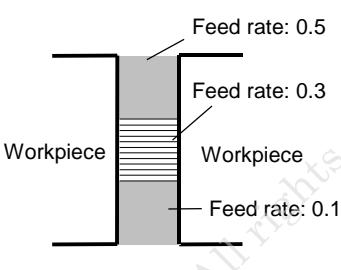
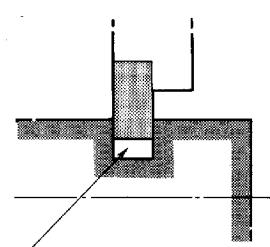
Dmax: Stock material maximum outside diameter
 Dmin: Stock material minimum inside diameter
 l_0 : Stock material edge projection length
 l : Stock material length

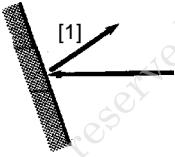
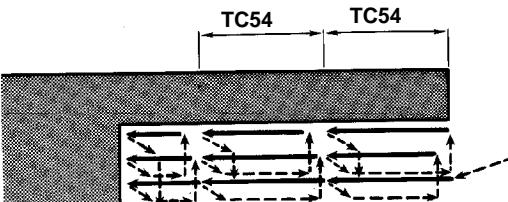
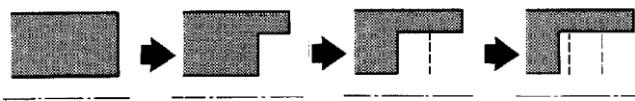
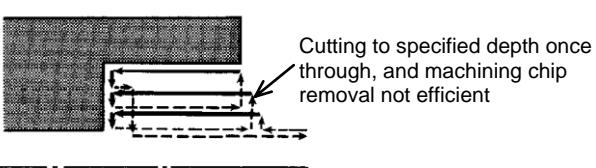
Classification	USER	Display title	TURNING								
TC41		<p>Address</p> <p>Meaning</p> <p>Thread cutting clearance (radial value)</p>	<p>Description</p> <p>Thread cutting clearance is provided to specify tool return distance for each cycle in thread cutting unit. Thread cutting clearance will be added to the highest portion of thread and repeating path will be determined automatically.</p> <p><OUT></p>  <p>TC41</p> <p>Programmed shape</p> <p>Thread cutting acceleration distance</p> <p>NM211-00222</p> <p><IN></p>  <p>Programmed shape</p> <p>Thread cutting acceleration distance</p> <p>TC41</p> <p>NM211-00223</p> <p><FCE></p>  <p>Programmed shape</p> <p>TC41</p> <p>Thread cutting acceleration distance</p> <p>NM211-00224</p> <p><BAK></p>  <p>Thread cutting acceleration distance</p> <p>TC41</p> <p>Programmed shape</p> <p>NM211-00225</p>								
		<table border="1"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>0.001 mm/0.0001 in</td> </tr> <tr> <td>Setting range</td> <td>0 to 65535</td> </tr> </table>	Program type	M	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535	
Program type	M										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										

Classification	USER	Display title	TURNING								
Address	Meaning	Description									
TC42	<p>Groove cutting clearance (radial value) in X-axis</p> <table border="1" style="margin-left: 10px;"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535	<p>Groove cutting clearance is provided at machining start portion in groove cutting unit.</p> <p><OUT></p>  <p><IN></p>  <p><FCE></p> 	<p>NM211-00226</p> <p>NM211-00227</p> <p>NM211-00228</p> <p>NM211-00229</p>
Program type	M										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										
TC43	<p>Groove cutting clearance in Z-axis</p> <table border="1" style="margin-left: 10px;"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535	<p><BAK></p> 	
Program type	M										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										

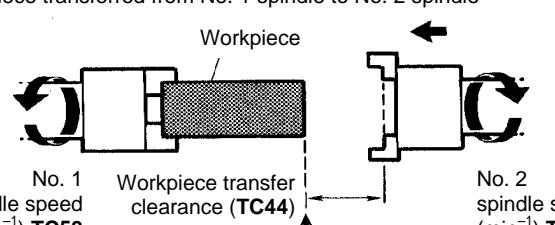
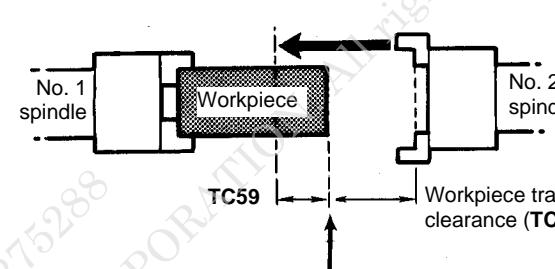
Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC44		<p>Workpiece transfer clearance is provided to specify workpiece transfer position in workpiece transfer unit.</p> 	
TC44	Workpiece transfer clearance	<p>No. 2 spindle traverses by rapid feed rate to a position distant from transfer position by clearance distance, and then transfer operation is initiated.</p> <p>(⇒ TC57, TC58, TC59)</p>	NM211-00231
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 65535	
TC45	Amount of the edge clearance after edge-machining	<p>After edge-machining unit, this parameter works instead of safety contour clearance FCE parameter TC39.</p> <p>If, however, TC45 is zero, then TC39 is used.</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 65535	
TC46	Drilling depth decrement in turning-drilling unit	 <p>D : Drilling depth d_1 : Cut depth in 1st cycle d_i : Cut depth in i-th cycle $d_1 - TC46 \times (i - 1)$ ($d_i \geq b$) b ($d_i < b$) b : Drilling depth clamping value (TC48)</p>	NM211-00251
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 65535	
(⇒ TC48)			

Classification	USER	Display title	TURNING	
Address	Meaning	Description		
TC47	Pecking return distance in turning-drilling unit	 P ₁ : Start point P ₂ : End point D _n (n = 1 to 4) = Cut depth		
	Program type	M	NM211-00252	
	Conditions	Immediate		
	Unit	0.001 mm/0.0001 in		
	Setting range	0 to 65535		
TC48	Drilling cut depth clamp value in turning-drilling unit	Minimum turning-drilling cut depth is set.		
	Program type	M	(⇒ TC46)	
	Conditions	Immediate		
	Unit	0.001 mm/0.0001 in		
	Setting range	0 to 65535		
TC49	Spindle speed clamp value in cutting off cycle (GRV)	If 0 or 1 is set up in TC50 : This value will be used as the spindle speed clamp value in cutting off cycle (#4/#5). If 2 or more is set up in TC50 : Immediately before the cutting off area (*) is reached during the cutting off cycle, this value will be used as the spindle speed clamp value. Within the cutting off area, however, machining will be executed at the speed corresponding to this value. *: Cutting off area = (Starting position X – Ending position X) × TC9/100		
	Program type	M		
	Conditions	Immediate		
	Unit	min ⁻¹		
	Setting range	0 to 65535		
		TC49 = 0: The spindle speed will be clamped with the RPM value in the common unit. If RPM is zero, the spindle speed will be clamped with the value of parameter SA1 .		

Classification	USER	Display title	TURNING								
Address	Meaning	Description									
TC50	<p>Number of times that the feed rate is to be reduced during the #4 and #5 cutting-off cycles of a grooving unit</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Times</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Times	Setting range	0 to 65535	<p>The starting feed value for cutting-off is a feed value that has been designated in unit data, and the ending feed value for cutting-off is a feed value that has been designated in sequence data. The feed rate from the start of machining to the end is reduced in steps according to the number of times that has been designated here.</p> <p>Example :</p> <p>Feed rate set at feed item in unit data = 0.5 Feed rate set at roughness item in sequence data = 0.1 TC50 = 3</p>  <p>When 0 or 1 is set for TC50, the feed rate is unchanged.</p>	
Program type	M										
Conditions	Immediate										
Unit	Times										
Setting range	0 to 65535										
TC51	<p>Dwell at the hole bottom during non-through hole drilling cycle of the turning-drilling unit</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Revolutions</td></tr> <tr><td>Setting range</td><td>0 to 255</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Revolutions	Setting range	0 to 255	<p>Set the number of revolutions of the spindle during dwell time at the bottom of a hole when the #0 - #4 non-through hole drilling cycle is selected in the turning-drilling unit.</p>	
Program type	M										
Conditions	Immediate										
Unit	Revolutions										
Setting range	0 to 255										
TC52	<p>Dwell (specification of spindle rotation number) at groove bottom in groove cutting unit</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Revolutions</td></tr> <tr><td>Setting range</td><td>0 to 255</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Revolutions	Setting range	0 to 255	<p>Tool will stop at groove bottom while spindle rotates N times when TC52 is set to N (N=0 to 255).</p>  <p>Remaining at groove bottom until the spindle rotates N times.</p> <p>NM211-00218</p>	
Program type	M										
Conditions	Immediate										
Unit	Revolutions										
Setting range	0 to 255										

Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC53	Feed rate for escape by short distance		For escape by very short distance, G01 feed speed will be faster than G00 to complete the operation. (If G00 is used, smoothing 0 detection will be made at the end position.) Therefore, for escape very short distance, use G01 command, and set the feed speed of this command as parameter.
			Example :
	Escape in finish cutting of edge-machining unit		Escape in rough cutting of bar machining unit
			
	<p>[1]: Feed speed in this block specified by G01 (TC53)</p> <p>NM211-00238</p>		
TC54	Program type	M	(⇒ TC67, TC68)
	Conditions	Immediate	
	Unit	mm/min / 0.1 in/min	
	Setting range	0 to 65535	
	Cut depth per cycle for machining inside diameter in bar machining unit	Inside diameter enlarging cycle	
			<p>Cutting is promoted gradually from the edge, and machining chip removal efficient.</p> 
			cf. Standard inside diameter cutting
			
			NM211-00242
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 65535	

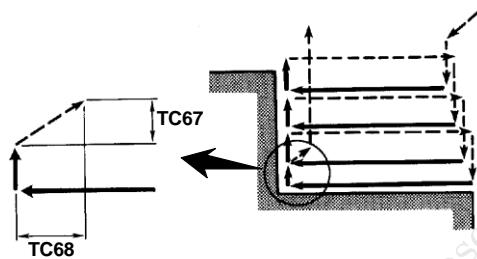
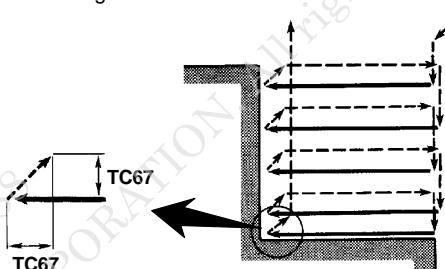
Classification	USER	Display title	TURNING								
Address	Meaning	Description									
TC55	<p>Reverse feed tolerance for contour machining</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535	<p>Example : Outside diameter machining in normal (- Z-axis) direction</p> <p>dr: Reverse feed contour data</p> <p>Contour</p> <p>dr</p> <p>$dr \leq TC55 \dots\dots\dots$ No alarm $dr > TC55 \dots\dots\dots$ Alarm</p> <p>NM211-00243</p>	
Program type	M										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										
TC56	<p>Overtravelling in X-axis direction in edge machining unit</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535	<p>P_S: Start point P_E: End point</p> <p>TC56</p> <p>NM211-00244</p> <p>Note : By setting an adequate value for TC56, uncut residue will not be produced in edge machining.</p> <p>Tool</p> <p>Uncut residue because of nose R. etc.</p> <p>NM211-00245</p>	
Program type	M										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										
TC57	<p>Workpiece pressing speed in workpiece transfer unit</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>mm/min 0.1 in/min</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	mm/min 0.1 in/min	Setting range	0 to 65535	<p>Example : Workpiece transferred from No. 1 spindle to No. 2 spindle</p> <p>Workpiece</p> <p>No. 1 spindle</p> <p>No. 2 spindle</p> <p>TC57</p> <p>Workpiece transfer clearance (TC44)</p> <p>Workpiece transfer position specified in transfer display</p> <p>NM211-00236</p> <p>Set this parameter equal to or as close as possible to its standard value. Excessive setting may cause a contact error. (⇒TC44, TC59)</p>	
Program type	M										
Conditions	Immediate										
Unit	mm/min 0.1 in/min										
Setting range	0 to 65535										

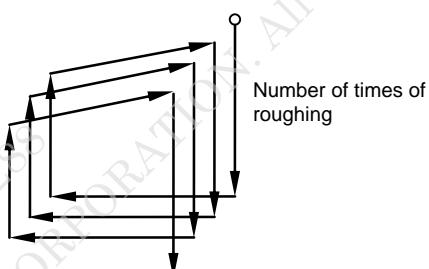
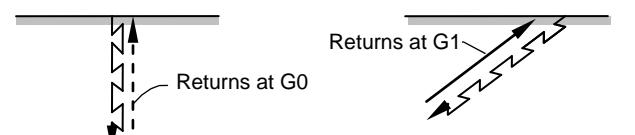
Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC58	Spindle speed (min^{-1}) of two spindles in workpiece transfer while the spindles are rotating in workpiece transfer unit		Workpiece transferred from No. 1 spindle to No. 2 spindle
	Program type	M	 <p>Workpiece transfer position specified in the transfer display</p> <p>NM211-00237</p>
	Conditions	Immediate	
	Unit	min^{-1}	
	Setting range	0 to 65535	
TC59	Workpiece pressing distance in workpiece transfer unit		Workpiece transferred from No. 1 spindle to No. 2 spindle
	Program type	M	 <p>Workpiece transfer position specified in the transfer display</p> <p>NM211-00255</p>
	Conditions	Immediate	
	Unit	0.001 mm/min 0.0001 in/min	
	Setting range	0 to 65535	
TC61	Simultaneous operation pattern for transfer		<p>Simultaneous operation pattern for transfer of workpieces between two unit jobsites</p> <p>TC61 = 1: Rotation of the spindle and movement of the Z-axis</p> <p>TC61 = 2: Orientation of the spindle and movement of the Z-axis</p> <p>TC61 = 4: Positioning of the C-axis and movement of the Z-axis</p> <p>Note : To combine patterns, set the sum total of setting numbers corresponding the conditions.</p>
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 7	

Classification	USER	Display title	TURNING																																																																								
Address	Meaning	Description																																																																									
TC62	Selection of tool change command output position for FLASH tool	<p>When a FLASH tool is used, specify the position for indexing the tool, namely, the tool change command output position for indexing the tool of the same TNo. at the same B-axis angle during programmed operation. In all other cases, the tool change command output position specified by SU10 becomes valid.</p> <table border="1"> <thead> <tr> <th>Setting</th> <th>X-axis</th> <th>Z-axis</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>Clearance position</td> <td>Clearance position</td> </tr> <tr> <td>1</td> <td>Machine zero point</td> <td>Clearance position</td> </tr> <tr> <td>2</td> <td>Clearance position</td> <td>Machine zero point</td> </tr> <tr> <td>3</td> <td>Machine zero point</td> <td>Machine zero point</td> </tr> <tr> <td>4</td> <td>Fixed point</td> <td>Fixed point</td> </tr> <tr> <td>5</td> <td>Clearance position</td> <td>End point of previous machining</td> </tr> <tr> <td>6</td> <td>Machine zero point</td> <td>End point of previous machining</td> </tr> <tr> <td>7</td> <td>End point of previous machining</td> <td>Clearance position</td> </tr> <tr> <td>8</td> <td>End point of previous machining</td> <td>Machine zero point</td> </tr> <tr> <td>9</td> <td>User-specified fixed point 1</td> <td>User-specified fixed point 1</td> </tr> <tr> <td>10</td> <td>User-specified fixed point 2</td> <td>User-specified fixed point 2</td> </tr> </tbody> </table> <p>Note :</p> <ol style="list-style-type: none"> 1. TC62 = 5 or 6, Z-axis tool change command output position is identical with the end point of previous machining. In the case below, however, this may not be applied. As shown here, if the longest tool comes into the hatched portion, the position will escape in Z-axis direction by the distance determined by SU51. 2. The turret type of turret conducts tool turning at the tool change command output position. The ATC type of turret moves from the tool change command output position to the ATC position. 	Setting	X-axis	Z-axis	0	Clearance position	Clearance position	1	Machine zero point	Clearance position	2	Clearance position	Machine zero point	3	Machine zero point	Machine zero point	4	Fixed point	Fixed point	5	Clearance position	End point of previous machining	6	Machine zero point	End point of previous machining	7	End point of previous machining	Clearance position	8	End point of previous machining	Machine zero point	9	User-specified fixed point 1	User-specified fixed point 1	10	User-specified fixed point 2	User-specified fixed point 2	<p>When a FLASH tool is used, specify the position for indexing the tool, namely, the tool change command output position for indexing the tool of the same TNo. at the same B-axis angle during programmed operation. In all other cases, the tool change command output position specified by SU10 becomes valid.</p> <table border="1"> <thead> <tr> <th>Setting</th> <th>X-axis</th> <th>Z-axis</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>Clearance position</td> <td>Clearance position</td> </tr> <tr> <td>1</td> <td>Machine zero point</td> <td>Clearance position</td> </tr> <tr> <td>2</td> <td>Clearance position</td> <td>Machine zero point</td> </tr> <tr> <td>3</td> <td>Machine zero point</td> <td>Machine zero point</td> </tr> <tr> <td>4</td> <td>Fixed point</td> <td>Fixed point</td> </tr> <tr> <td>5</td> <td>Clearance position</td> <td>End point of previous machining</td> </tr> <tr> <td>6</td> <td>Machine zero point</td> <td>End point of previous machining</td> </tr> <tr> <td>7</td> <td>End point of previous machining</td> <td>Clearance position</td> </tr> <tr> <td>8</td> <td>End point of previous machining</td> <td>Machine zero point</td> </tr> <tr> <td>9</td> <td>User-specified fixed point 1</td> <td>User-specified fixed point 1</td> </tr> <tr> <td>10</td> <td>User-specified fixed point 2</td> <td>User-specified fixed point 2</td> </tr> </tbody> </table> <p>Note :</p> <ol style="list-style-type: none"> 1. TC62 = 5 or 6, Z-axis tool change command output position is identical with the end point of previous machining. In the case below, however, this may not be applied. As shown here, if the longest tool comes into the hatched portion, the position will escape in Z-axis direction by the distance determined by SU51. 2. The turret type of turret conducts tool turning at the tool change command output position. The ATC type of turret moves from the tool change command output position to the ATC position. 	Setting	X-axis	Z-axis	0	Clearance position	Clearance position	1	Machine zero point	Clearance position	2	Clearance position	Machine zero point	3	Machine zero point	Machine zero point	4	Fixed point	Fixed point	5	Clearance position	End point of previous machining	6	Machine zero point	End point of previous machining	7	End point of previous machining	Clearance position	8	End point of previous machining	Machine zero point	9	User-specified fixed point 1	User-specified fixed point 1	10	User-specified fixed point 2	User-specified fixed point 2
Setting	X-axis	Z-axis																																																																									
0	Clearance position	Clearance position																																																																									
1	Machine zero point	Clearance position																																																																									
2	Clearance position	Machine zero point																																																																									
3	Machine zero point	Machine zero point																																																																									
4	Fixed point	Fixed point																																																																									
5	Clearance position	End point of previous machining																																																																									
6	Machine zero point	End point of previous machining																																																																									
7	End point of previous machining	Clearance position																																																																									
8	End point of previous machining	Machine zero point																																																																									
9	User-specified fixed point 1	User-specified fixed point 1																																																																									
10	User-specified fixed point 2	User-specified fixed point 2																																																																									
Setting	X-axis	Z-axis																																																																									
0	Clearance position	Clearance position																																																																									
1	Machine zero point	Clearance position																																																																									
2	Clearance position	Machine zero point																																																																									
3	Machine zero point	Machine zero point																																																																									
4	Fixed point	Fixed point																																																																									
5	Clearance position	End point of previous machining																																																																									
6	Machine zero point	End point of previous machining																																																																									
7	End point of previous machining	Clearance position																																																																									
8	End point of previous machining	Machine zero point																																																																									
9	User-specified fixed point 1	User-specified fixed point 1																																																																									
10	User-specified fixed point 2	User-specified fixed point 2																																																																									
Program type	M																																																																										
Conditions	Immediate																																																																										
Unit	—																																																																										
Setting range	0 to 10																																																																										

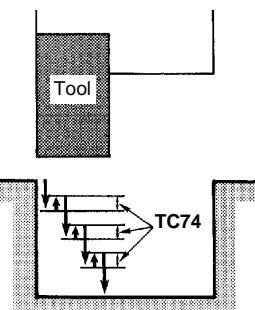
Classification	USER	Display title	TURNING																																							
Address		Meaning	Description																																							
		Selection of tool change command output position for FLASH tool	When a FLASH tool is used, specify the position for indexing the tool, namely, the tool change command output position for indexing the tool of the same TNo. at the same B-axis angle during programmed operation. In all other cases, the tool change position specified by SU10 becomes valid.																																							
		INTEGREX e-H	<p>The diagram shows a machine setup with two main horizontal axes: X-axis and Z-axis. There are three zero points: 2nd zero point, Machine zero point, and 3rd zero point. Tools are indexed between these points. Labels include M6, M5, I21/I22, Dmax, Zc, beta_x, and beta_z. A table below lists settings for TC62 from 0 to 11, mapping them to specific tool change positions.</p> <table border="1"> <thead> <tr> <th>Setting</th> <th>X-axis</th> <th>Z-axis</th> </tr> </thead> <tbody> <tr><td>0</td><td>Clearance position</td><td>Clearance position</td></tr> <tr><td>1</td><td>Machine zero point</td><td>Clearance position</td></tr> <tr><td>2</td><td>Clearance position</td><td>Machine zero point</td></tr> <tr><td>3</td><td>Machine zero point</td><td>Machine zero point</td></tr> <tr><td>4</td><td>2nd zero point</td><td>2nd zero point</td></tr> <tr><td>5</td><td>Clearance position</td><td>End point of previous machining</td></tr> <tr><td>6</td><td>Machine zero point</td><td>End point of previous machining</td></tr> <tr><td>7</td><td>End point of previous machining</td><td>Clearance position</td></tr> <tr><td>8</td><td>End point of previous machining</td><td>Machine zero point</td></tr> <tr><td>9</td><td>User-specified fixed point 1</td><td>User-specified fixed point 1</td></tr> <tr><td>10</td><td>User-specified fixed point 2</td><td>User-specified fixed point 2</td></tr> <tr><td>11</td><td>3rd zero point</td><td>End point of previous machining</td></tr> </tbody> </table>	Setting	X-axis	Z-axis	0	Clearance position	Clearance position	1	Machine zero point	Clearance position	2	Clearance position	Machine zero point	3	Machine zero point	Machine zero point	4	2nd zero point	2nd zero point	5	Clearance position	End point of previous machining	6	Machine zero point	End point of previous machining	7	End point of previous machining	Clearance position	8	End point of previous machining	Machine zero point	9	User-specified fixed point 1	User-specified fixed point 1	10	User-specified fixed point 2	User-specified fixed point 2	11	3rd zero point	End point of previous machining
Setting	X-axis	Z-axis																																								
0	Clearance position	Clearance position																																								
1	Machine zero point	Clearance position																																								
2	Clearance position	Machine zero point																																								
3	Machine zero point	Machine zero point																																								
4	2nd zero point	2nd zero point																																								
5	Clearance position	End point of previous machining																																								
6	Machine zero point	End point of previous machining																																								
7	End point of previous machining	Clearance position																																								
8	End point of previous machining	Machine zero point																																								
9	User-specified fixed point 1	User-specified fixed point 1																																								
10	User-specified fixed point 2	User-specified fixed point 2																																								
11	3rd zero point	End point of previous machining																																								
		INTEGREX e-V	<p>The diagram shows a similar machine setup to e-H, but with a different tool indexing sequence. It includes labels I21/I22, M6, M5, X-axis, Z-axis, beta_x, beta_z, Dmax, Zc, and 2nd zero point. A note specifies that setting TC62 = 4 is not recommended for e-II series due to potential collisions.</p>																																							
		Program type	M																																							
		Conditions	Immediate																																							
		Unit	—																																							
		Setting range	0 to 11																																							
		<p>A diagram showing a hatched area representing stock material. Labels include SU51, Stock material length, and Stock material edge protrusion length.</p>																																								

Classification	USER	Display title	TURNING								
Address	Meaning	Description									
TC63	<p>Amount of relief after transfer using the workpiece transfer unit (Spindle mode 0 to 5)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>0.1 mm/0.01 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	After stop of movement	Unit	0.1 mm/0.01 in	Setting range	0 to 65535	<p>Set the amount of relief after executing the workpiece transfer unit. Example : Transfer from HD2 to HD1</p>	
Program type	M										
Conditions	After stop of movement										
Unit	0.1 mm/0.01 in										
Setting range	0 to 65535										
TC64	<p>Amount of relief after transfer using the workpiece transfer unit (Spindle mode 6 and 7)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>0.01 s</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	After stop of movement	Unit	0.01 s	Setting range	0 to 65535		
Program type	M										
Conditions	After stop of movement										
Unit	0.01 s										
Setting range	0 to 65535										
TC65	<p>Specification of first M-code for parts catcher control</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 255</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	—	Setting range	0 to 255	<p>It is a parameter to automatically control the parts catcher. If the set value of TC65 is n, M-code of No. n (parts catcher forward) is outputted at the start of cutting off (#4, #5), and M-code of No. n+1 (Parts catcher backward) is outputted at the end.</p> <p>Note : If 0 is set in TC65 no M-code is outputted.</p>	
Program type	M										
Conditions	Immediate										
Unit	—										
Setting range	0 to 255										
TC66	<p>Minimum index angle of the FLASH tool</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.1°</td></tr> <tr><td>Setting range</td><td>0 to 3599</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.1°	Setting range	0 to 3599	<p>This parameter is used as the basis for judging whether the index angle entered in TOOL DATA display is acceptable. An alarm will be displayed if the entered value is judged to be unacceptable. (Specify "900" for 4-segment splitting.)</p>	
Program type	M, E										
Conditions	Immediate										
Unit	0.1°										
Setting range	0 to 3599										

Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC67	Return distance (radial value) in X-axis at wall during rough cutting in bar cutting unit, in corner machining unit or in mill-turning unit of MAZATROL programs		Example : 1. Standard type cutting Prior to cutting up along the wall in the end of final cycle, escape will be made by specified distance. 
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 65535	
TC68	Return distance in Z-axis at wall during rough cutting in bar cutting unit, in corner machining unit or in mill-turning unit of MAZATROL programs		2. High speed rough cutting Escape will be made by TC67 and TC68 specified distance during return after reaching the wall. 
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 65535	
TC69	Number of revolutions during dwell for each cutting operation for machining #0, #1, #2, #3, #4, or #5 groove (T. GROOVE unit) with pecking return distance (TC74) of 0.		Set number of revolutions during dwell for each cutting operation for machining #0, #1, #2, #3, #4, or #5 groove (T. GROOVE unit) with pecking return distance (TC74) of 0. The tool stops moving until the spindle makes revolutions set in this parameter.
	Program type	M	
	Conditions	Immediate	
	Unit	Revolutions	
	Setting range	0 to 255	
TC70	FLASH tool — Number of cutting edges to be used for the tool not registered in the tool file		Number of cutting edges to be used if the tool file data for the corresponding tool is not yet registered during FLASH tool selection.
	Program type	M	
	Conditions	Immediate	
	Unit	Number of cutting edges	
	Setting range	0 to 99	

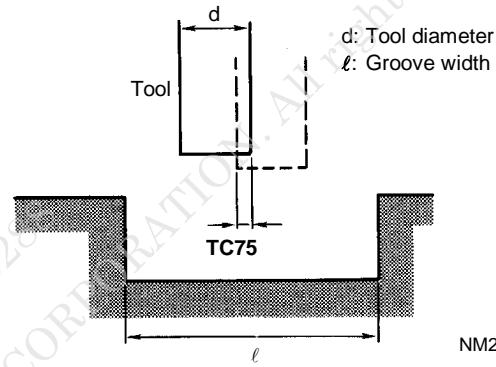
Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC71	Feed stopping rotation dwell time during the chip cutting cycle (valid only for roughing)	<p>Specify the feed stopping rotation dwell time during the chip cutting cycle. Feed is stopped while the spindle rotates for the specified time.</p> <p>Note : If "0" is entered, the dwell time will be handled as "0". However, the feed rate will be reduced since the dwell function itself will be executed.</p>	
	Program type M Conditions Immediate Unit Revolutions Setting range 0 to 65535		
TC72	Number of times of roughing in the composite-type fixed cycle (G273, G73)	<p>If the number of times of roughing has not been specified in the program, operation will occur in accordance with the setting of this parameter.</p> 	
	Program type E Conditions Immediate Unit Times Setting range 0 to 65535		
TC73	Return speed at pecking portion in groove cutting unit and turning-drilling unit	<p>Pecking speed</p> <p>[1] Cutting: (G1) F command data [2] Pecking: (G1) TC73 [3] Cutting: (G1) F command data</p> <p>When executing groove cutting with grooving pattern #0 (only for oblique groove) the tool returns from the groove bottom also at the "G1" feed rate determined by this parameter.</p> <p>Grooving #0 (Right-angled groove) Grooving #0 (Oblique groove)</p>  <p>Note : In turning-drilling unit, TC73 setting will be used only for #2 and [#2] types.</p>	
	Program type M Conditions Immediate Unit 0.001 mm/rev 0.0001 in/rev Setting range 0 to 65535		

Classification	USER	Display title	TURNING
Address		Meaning	
TC74		Pecking return distance in groove cutting unit of MAZATROL programs	
		Program type	M
		Conditions	Immediate
		Unit	0.001 mm/0.0001 in
		Setting range	0 to 65535
TC75		Overlap distance for machining wide groove in groove cutting unit	
		Program type	M
		Conditions	Immediate
		Unit	0.001 mm/0.0001 in
		Setting range	0 to 65535
TC76		Escape value after machining in edge machining unit	
		Program type	M
		Conditions	Immediate
		Unit	0.001 mm/0.0001 in
		Setting range	0 to 65535
TC77		Acceleration distance clamp value for thread cutting unit	
		Program type	M
		Conditions	Immediate
		Unit	Lead/10
		Setting range	0 to 255

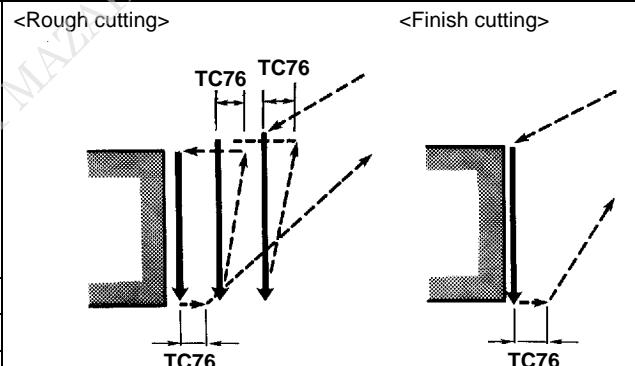


* For EIA/ISO programs (fixed cycle) refer to **SU104**.

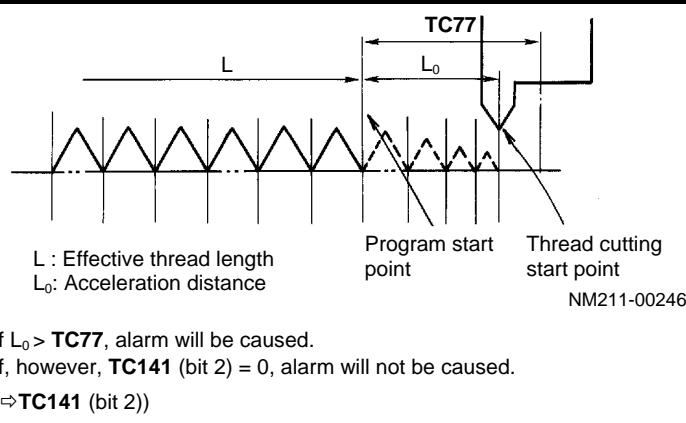
NM211-00248

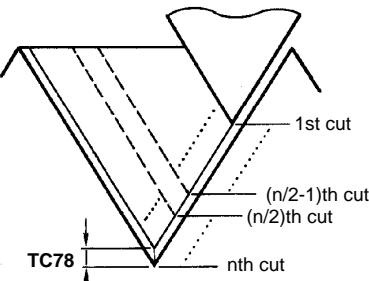
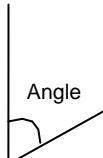
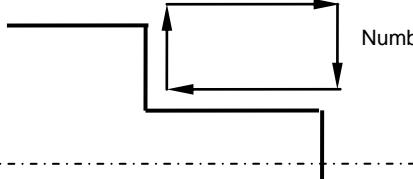


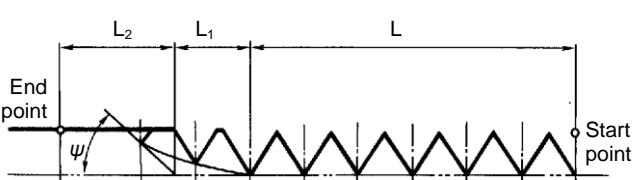
NM211-00249



NM211-00250



Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC78	Cut depth (radial value) for final cut in thread cutting unit of MAZATROL programs	 <small>* For EIA/ISO programs (fixed cycle) refer to SU105.</small>	NM211-00247
TC79	Minimum cut depth clamping value (radial value) in thread cutting unit of MAZATROL programs	Clamping will follow the setting of TC79 if the calculated value of the cut depth with the threading unit is smaller than the setting of TC79 . This parameter is valid only for the infeed operation of the fixed-area scheme. <small>* For EIA/ISO programs (fixed cycle) refer to SU106.</small>	
TC80	Angle of the tool nose during the G276/G76 mode	If, during the composite-type fixed cycle G276/G76 mode, a tool nose angle has not been specified in the program, the setting of this parameter will become the angle of the tool nose.  The setting must be either 0, 29, 30, 55, 60 or 80.	
TC81	Final finishing repeat times in the composite-type fixed cycle (G276, G76)	If the number of times of repetition has not been specified in the program, operation will occur in accordance with the setting of this parameter. 	
	Program type	E	
	Conditions	Immediate	
	Unit	°	
	Setting range	0, 29, 30, 55, 60, 80	
	Program type	E	
	Conditions	Immediate	
	Unit	Times	
	Setting range	0 to 65535	

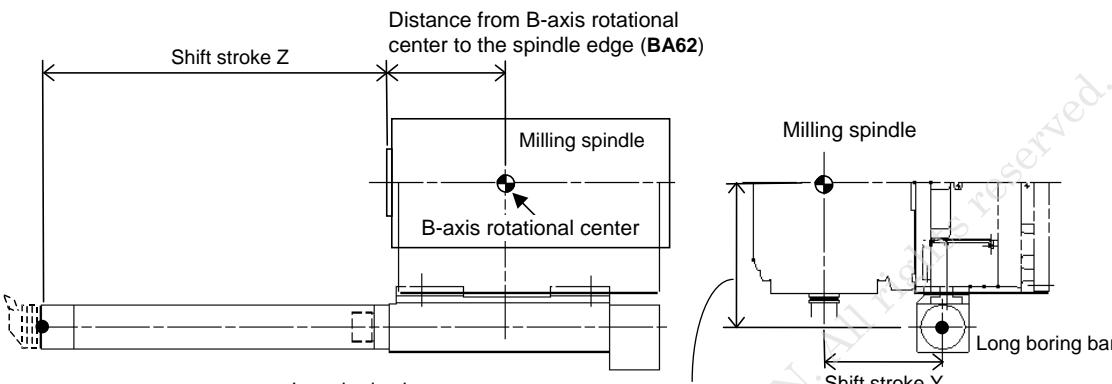
Classification	USER	Display title	TURNING								
Address	Meaning	Description									
TC82	<p>Chamfering data calculation coefficient in thread cutting unit and thread cutting cycle (G276/G292, G76/G92)</p> <table border="1" data-bbox="285 1010 682 1156"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Lead/10</td></tr> <tr><td>Setting range</td><td>0 to 40</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	Lead/10	Setting range	0 to 40	 <p>NM211-00272</p> <p> L : Effective thread length L_1 : Same pitch incomplete thread length (follow-up delay) L_2 : Chamfering data ψ : Chamfering angle </p> $L_2 = L_0 \times \frac{TC82}{10}$ <p>L_0 : Thread lead</p>	
Program type	M, E										
Conditions	Immediate										
Unit	Lead/10										
Setting range	0 to 40										
TC83	<p>Number of cutting operations to be performed on finishing allowance corresponding to standard pattern (#0) of threading unit</p> <table border="1" data-bbox="285 1841 682 1998"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Times</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Times	Setting range	0 to 65535	<p>During a finishing process based on the standard pattern (#0) of the threading unit, TC78 cutting (final cutting diameter in threading unit) is repeated the number of times that has been specified in TC83.</p> <p>TC83 = 0 or 1: Cutting based on the setting of TC78 occurs once.</p> <p>TC83 ≥ 2: Finish-cutting is repeated the number of times specified in TC83, and with the depth-of-cut setting of TC78/TC83.</p> <p><Supplementary description></p> <ul style="list-style-type: none"> Parameter TC83 is valid only for #0, [#0]: it does not function for #1, [#1] or #2, [#2]. If TC78 = 0, TC83 is valid. <p>For thread refinishing, one cutting operation is performed as before.</p>	
Program type	M										
Conditions	Immediate										
Unit	Times										
Setting range	0 to 65535										

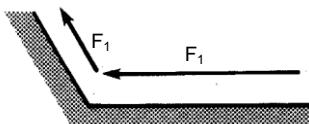
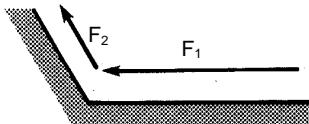
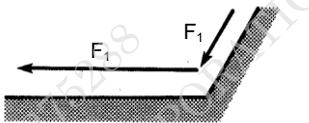
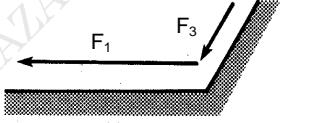
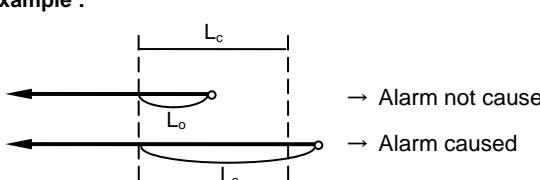
Classification	USER	Display title	TURNING																						
Address	Meaning	Description																							
TC84	Feed rate to be auto-set for finishing Program type M Conditions Immediate Unit 0.001 mm/rev / 0.0001 in/rev Setting range 0 to 65535	Specify the feed rate to be auto-set for finishing (turning) in the MAZATROL program.																							
TC85 to TC94	Specification of the pocket for the long boring bar Program type M, E Conditions Immediate Unit — Setting range 0 to 960	Specify the number of the magazine pocket holding the adapter for the long boring bar. <table border="1"> <thead> <tr> <th>Address</th><th>Description</th></tr> </thead> <tbody> <tr><td>TC85</td><td>Pocket No. 1 for long boring bar</td></tr> <tr><td>TC86</td><td>Pocket No. 2 for long boring bar</td></tr> <tr><td>TC87</td><td>Pocket No. 3 for long boring bar</td></tr> <tr><td>TC88</td><td>Pocket No. 4 for long boring bar</td></tr> <tr><td>TC89</td><td>Pocket No. 5 for long boring bar</td></tr> <tr><td>TC90</td><td>Pocket No. 6 for long boring bar</td></tr> <tr><td>TC91</td><td>Pocket No. 7 for long boring bar</td></tr> <tr><td>TC92</td><td>Pocket No. 8 for long boring bar</td></tr> <tr><td>TC93</td><td>Pocket No. 9 for long boring bar</td></tr> <tr><td>TC94</td><td>Pocket No. 10 for long boring bar</td></tr> </tbody> </table>	Address	Description	TC85	Pocket No. 1 for long boring bar	TC86	Pocket No. 2 for long boring bar	TC87	Pocket No. 3 for long boring bar	TC88	Pocket No. 4 for long boring bar	TC89	Pocket No. 5 for long boring bar	TC90	Pocket No. 6 for long boring bar	TC91	Pocket No. 7 for long boring bar	TC92	Pocket No. 8 for long boring bar	TC93	Pocket No. 9 for long boring bar	TC94	Pocket No. 10 for long boring bar	
Address	Description																								
TC85	Pocket No. 1 for long boring bar																								
TC86	Pocket No. 2 for long boring bar																								
TC87	Pocket No. 3 for long boring bar																								
TC88	Pocket No. 4 for long boring bar																								
TC89	Pocket No. 5 for long boring bar																								
TC90	Pocket No. 6 for long boring bar																								
TC91	Pocket No. 7 for long boring bar																								
TC92	Pocket No. 8 for long boring bar																								
TC93	Pocket No. 9 for long boring bar																								
TC94	Pocket No. 10 for long boring bar																								
TC95 TC96	— Program type — Conditions — Unit — Setting range —	Fixed value																							
TC97	Type of retraction during workpiece transfer Program type M Conditions Immediate Unit — Setting range 0 to 2	Specify the sequence of retraction during workpiece transfer on machines with the lower turret. TC97 = 0: The upper and lower turrets return simultaneously. TC97 = 1: The upper turret returns prior to the lower turret. TC97 = 2: The lower turret returns prior to the upper turret. The setting is regarded as "0" if out of range.																							

Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC98	Returning operation after machining specified in the END unit	<p>Specify the sequence of the returning operation set in the ATC and RETURN items in the end unit for machines with the lower turret.</p> <p>TC98 = 0: The upper and lower turrets return simultaneously.</p> <p>TC98 = 1: The upper turret returns prior to the lower turret.</p> <p>TC98 = 2: The lower turret returns prior to the upper turret.</p> <p>The setting is regarded as "0" if out of range.</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	
TC99	ATC operation after machining when not specified in the END unit	<p>Specify the ATC operation after machining when the ATC item in the end unit is not set.</p> <p>TC99 = 0: The same operation as when the item is set to "0" is followed. (The tool is not returned.)</p> <p>TC99 = 1: The same operation as when the item is set to "1" is followed. (The tool is returned and axes move to the returning position.)</p> <p>TC99 = 2: The same pattern as when the item is set to "2" is followed. (The axes move to the returning position and the tool is returned).</p> <p>The setting is regarded as "0" if out of range.</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	
TC101	Selection of droop sampling axis (For detecting imbalance)	Select a droop sampling axis.	
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 16	
TC102	Selection of cycle counter sampling axis (For detecting imbalance)	Select a cycle counter sampling axis.	
	Program type	E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 16	

Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC103	Amplitude limit of table vibration (For detecting imbalance)	Set the amplitude limit of table vibration.	
	Program type E		
	Conditions At power on		
	Unit —		
	Setting range 0 to 9999		
TC104	Clearance for rechucking using the workpiece transfer unit	<p>Set the clearance for rechucking the workpiece using the workpiece transfer unit.</p> <p>Movement to the clearance position occurs in the G0 mode.</p> <p>Movement from the clearance position to the gripping position occurs in the G1 mode using the feed rate specified in parameter TC57.</p> <ul style="list-style-type: none"> • Spindle No. 1 specification <p>Workpiece</p> <p>G0 → G1 → Gripping position</p> <p>TC104</p> <ul style="list-style-type: none"> • Spindle No. 2 specification <p>Gripping position ← G1 ← G0</p> <p>Workpiece</p> <p>TC104</p>	
TC106	Designation of MOVE C in a workpiece transfer unit of a program of workpiece scheme	<p>Select whether or not to enable the specification of MOVE C which becomes valid after the workpiece has been moved using the workpiece transfer unit (CHUCK/BAR/MOVE) in a program of workpiece scheme.</p> <p>0: Not to be enabled</p> <p>1: To be enabled</p> <p>MOVE C can be specified only when the workpiece transfer unit is set as follows: [CHUCK WORK] or [BAR LOOP] is selected for PAT. Either [3 ORIENT], [4 C-AXIS POSITION], or [5 KEEP POSITION] is selected for SPDL.</p> <p>Also, MOVE C needs to be specified in incremental data terms with respect to the value of C in the immediately preceding fundamental coordinate unit (WPC).</p> <p>(Only for the INTEGREX e-H Series)</p>	
	Program type M		
	Conditions Immediate		
	Unit Bit		
	Setting range 0, 1		

Classification	USER	Display title	TURNING								
Address	Meaning	Description									
TC111 to TC113	<p>CUTTING CONDITION LEARN display — Workpiece length range</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>mm/0.1 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	mm/0.1 in	Setting range	0 to 65535	<p>Specify the workpiece length range displayed on the CUTTING CONDITION LEARN display.</p> <p>Specify the range so that the relational expression of TC111 < TC112 < TC113 is established.</p>	
Program type	M										
Conditions	Immediate										
Unit	mm/0.1 in										
Setting range	0 to 65535										
TC114 to TC116	<p>CUTTING CONDITION LEARN display — Max. workpiece outside diameter range</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>mm/0.1 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	mm/0.1 in	Setting range	0 to 65535	<p>Specify the maximum outside diameter range of the workpieces displayed on the CUTTING CONDITION LEARN display.</p> <p>Specify the range so that the relational expression of TC114 < TC115 < TC116 is established.</p>	
Program type	M										
Conditions	Immediate										
Unit	mm/0.1 in										
Setting range	0 to 65535										
TC117	<p>Composite-type fixed cycle — amount of X-axial release (G273)</p> <table border="1"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	E	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535	The settings of these parameters will be used if the amount of release is not specified in the program.	
Program type	E										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										
TC118	<p>Composite-type fixed cycle — amount of Z-axial release (G273)</p> <table border="1"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	E	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 65535		
Program type	E										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 65535										

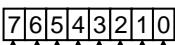
Classification	USER	Display title	TURNING																																				
Address	Meaning	Description																																					
TC120 to TC137	Distance to the front end of the long boring bar	Specify the distance from the B-axis rotational center of the milling spindle to the front end of the long boring bar. (On the 3D MONITOR display, long boring bar image is shown at the display position when the tool data is changed or the tool is changed to other tool after changing the parameter value.)																																					
		 <p>The diagram shows a vertical line representing the B-axis rotational center. A horizontal line represents the Milling spindle. A long horizontal bar represents the Long boring bar. Three arrows indicate shift strokes: Shift stroke Z (vertical arrow from the B-axis center to the spindle), Shift stroke X (horizontal arrow from the spindle to the front end of the long boring bar), and Shift stroke Y (vertical arrow from the spindle to the rear end of the long boring bar). The text "Distance from B-axis rotational center to the spindle edge (BA62)" is also present.</p>																																					
<table border="1"> <tr> <th>Address</th> <th>Description</th> </tr> <tr> <td>TC120</td> <td>Long boring bar No. 1 - Shift stroke X</td> </tr> <tr> <td>TC121</td> <td>Long boring bar No. 1 - Shift stroke Y</td> </tr> <tr> <td>TC122</td> <td>Long boring bar No. 1 - Shift stroke Z</td> </tr> <tr> <td>TC123</td> <td>Long boring bar No. 2 - Shift stroke X</td> </tr> <tr> <td>TC124</td> <td>Long boring bar No. 2 - Shift stroke Y</td> </tr> <tr> <td>TC125</td> <td>Long boring bar No. 2 - Shift stroke Z</td> </tr> <tr> <td>TC126</td> <td>Long boring bar No. 3 - Shift stroke X</td> </tr> <tr> <td>TC127</td> <td>Long boring bar No. 3 - Shift stroke Y</td> </tr> <tr> <td>TC128</td> <td>Long boring bar No. 3 - Shift stroke Z</td> </tr> <tr> <td>TC129</td> <td>Long boring bar No. 4 - Shift stroke X</td> </tr> <tr> <td>TC130</td> <td>Long boring bar No. 4 - Shift stroke Y</td> </tr> <tr> <td>TC131</td> <td>Long boring bar No. 4 - Shift stroke Z</td> </tr> <tr> <td>TC132</td> <td>Long boring bar No. 5 - Shift stroke X</td> </tr> <tr> <td>TC133</td> <td>Long boring bar No. 5 - Shift stroke Y</td> </tr> <tr> <td>TC134</td> <td>Long boring bar No. 5 - Shift stroke Z</td> </tr> <tr> <td>TC135</td> <td>Long boring bar No. 6 - Shift stroke X</td> </tr> <tr> <td>TC136</td> <td>Long boring bar No. 6 - Shift stroke Y</td> </tr> <tr> <td>TC137</td> <td>Long boring bar No. 6 - Shift stroke Z</td> </tr> </table>		Address	Description	TC120	Long boring bar No. 1 - Shift stroke X	TC121	Long boring bar No. 1 - Shift stroke Y	TC122	Long boring bar No. 1 - Shift stroke Z	TC123	Long boring bar No. 2 - Shift stroke X	TC124	Long boring bar No. 2 - Shift stroke Y	TC125	Long boring bar No. 2 - Shift stroke Z	TC126	Long boring bar No. 3 - Shift stroke X	TC127	Long boring bar No. 3 - Shift stroke Y	TC128	Long boring bar No. 3 - Shift stroke Z	TC129	Long boring bar No. 4 - Shift stroke X	TC130	Long boring bar No. 4 - Shift stroke Y	TC131	Long boring bar No. 4 - Shift stroke Z	TC132	Long boring bar No. 5 - Shift stroke X	TC133	Long boring bar No. 5 - Shift stroke Y	TC134	Long boring bar No. 5 - Shift stroke Z	TC135	Long boring bar No. 6 - Shift stroke X	TC136	Long boring bar No. 6 - Shift stroke Y	TC137	Long boring bar No. 6 - Shift stroke Z
Address	Description																																						
TC120	Long boring bar No. 1 - Shift stroke X																																						
TC121	Long boring bar No. 1 - Shift stroke Y																																						
TC122	Long boring bar No. 1 - Shift stroke Z																																						
TC123	Long boring bar No. 2 - Shift stroke X																																						
TC124	Long boring bar No. 2 - Shift stroke Y																																						
TC125	Long boring bar No. 2 - Shift stroke Z																																						
TC126	Long boring bar No. 3 - Shift stroke X																																						
TC127	Long boring bar No. 3 - Shift stroke Y																																						
TC128	Long boring bar No. 3 - Shift stroke Z																																						
TC129	Long boring bar No. 4 - Shift stroke X																																						
TC130	Long boring bar No. 4 - Shift stroke Y																																						
TC131	Long boring bar No. 4 - Shift stroke Z																																						
TC132	Long boring bar No. 5 - Shift stroke X																																						
TC133	Long boring bar No. 5 - Shift stroke Y																																						
TC134	Long boring bar No. 5 - Shift stroke Z																																						
TC135	Long boring bar No. 6 - Shift stroke X																																						
TC136	Long boring bar No. 6 - Shift stroke Y																																						
TC137	Long boring bar No. 6 - Shift stroke Z																																						
Program type	M, E																																						
Conditions	At power on																																						
Unit	0.0001 mm/0.00001 in																																						
Setting range	±99999999																																						

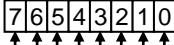
Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC141 (bit 0)	Use/disuse of acceleration in up-going slope during rough cutting cycle in bar machining unit	<p>TC141 (bit 0) = 0: Disuse</p>  <p>No acceleration</p> <p>TC141 (bit 0) = 1: Use</p>  <p>$F_2 > F_1$ F_1 : Feed rate for rough cutting cycle F_2 : Feed rate after acceleration</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	(⇒TC2, TC3) NM211-00208
TC141 (bit 1)	Use/disuse of deceleration in down-going slope during rough cutting cycle in bar machining unit	<p>TC141 (bit 1) = 0: Disuse</p>  <p>No deceleration</p> <p>TC141 (bit 1) = 1: Use</p>  <p>$F_3 < F_1$ F_1 : Feed rate for rough cutting cycle F_3 : Feed rate after deceleration</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	(⇒TC5, TC6) NM211-00209
TC141 (bit 2)	Selection between use/disuse of acceleration distance check at start of thread cutting unit	<p>TC141 (bit 2) = 0: Disuse</p> <p>Alarm not caused even if acceleration distance at start of thread cutting unit exceeds clamp data</p> <p>TC141 (bit 2) = 1: Use</p> <p>Alarm caused when acceleration distance at start of thread cutting unit exceeds clamp data</p> <p>Example :</p>  <p>Start of thread cutting</p> <p>L_o : Acceleration distance L_c : Acceleration distance clamp data</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	(⇒TC77)

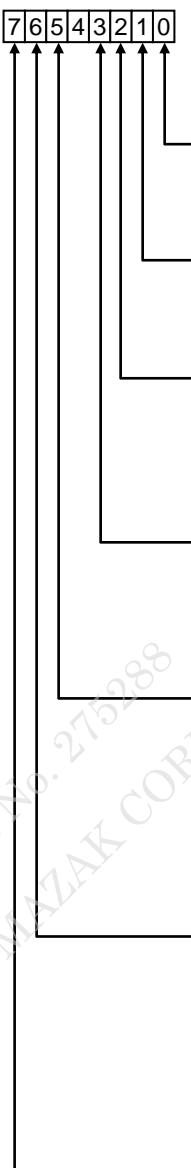
Classification	USER	Display title	TURNING																		
Address	Meaning	Description																			
TC141 (bit 3)	Selection between start position shift/start angle shift for thread number offset in thread cutting unit	<p>TC141 (bit 3) = 0: Start position shift Thread number offset adjusted by moving thread cutting start position</p> <p>TC141 (bit 3) = 1: Start angle shift Thread number offset adjusted according to thread cutting start angle</p>																			
TC141 (bit 4) (bit 5)	Selecting an angle margin for nose shape compensation	An angle margin for nose shape compensation can be selected by setting data in bits 4 and 5.	<table border="1"> <thead> <tr> <th colspan="2">Setting</th> <th>Angle margin for nose shape compensation</th> </tr> <tr> <th>Bit 5</th> <th>Bit 4</th> <th></th> </tr> </thead> <tbody> <tr> <td>0</td> <td>0</td> <td>3.0</td> </tr> <tr> <td>0</td> <td>1</td> <td>2.0</td> </tr> <tr> <td>1</td> <td>0</td> <td>1.0</td> </tr> <tr> <td>1</td> <td>1</td> <td>0.5</td> </tr> </tbody> </table>	Setting		Angle margin for nose shape compensation	Bit 5	Bit 4		0	0	3.0	0	1	2.0	1	0	1.0	1	1	0.5
Setting		Angle margin for nose shape compensation																			
Bit 5	Bit 4																				
0	0	3.0																			
0	1	2.0																			
1	0	1.0																			
1	1	0.5																			
TC141 (bit 6)	CHUCK JAW DATA display name/code selection	<p>TC141 (bit 6) = 1 A code can be selected.</p> <p>TC141 (bit 6) = 0 A name can be selected.</p>																			
TC141 (bit 7)	Whether to make the partition plate and the workpiece barrier valid	<p>TC141 (bit 7) = 1 Partition plate and the workpiece barrier valid</p> <p>TC141 (bit 7) = 0 Partition plate and the workpiece barrier invalid</p>																			

Classification	USER	Display title	TURNING
Address	Meaning		Description
TC142 (bit 0)	—		Fixed value (0)
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
TC142 (bit 1)	Selecting an inter-unit relief path when a succession of I.D. turning units using the same tool exist and there is no movement to the rotating position of the tool		<p>TC142 (bit 1) = 0 The relationship between the starting position of machining with the next unit and the ending position of machining with the previous unit is examined and if interference is judged to be likely, the tool will escape to a clearance point.</p> <p>TC142 (bit 1) = 1 The tool escapes to the clearance point each time the I.D. turning unit is executed to completion.</p>
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC142 (bit 2)	Selection of the jaw data reference method		<p>Specify the jaw data reference method.</p> <p>TC142 (bit 2) = 0: Reference using the code number of the jaw.</p> <p>TC142 (bit 2) = 1: Reference using the name of the jaw.</p>
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC142 (bit 3)	Using angle tool holder valid/invalid		<p>TC142 (bit 3) = 0: Angle tool holder cannot be used.</p> <p>TC142 (bit 3) = 1: Angle tool holder can be used.</p> <p>Use BA63 to set the angle of the angle tool holder.</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC142 (bit 5)	Selection whether an alarm is to be issued if the ending position of workpiece pressing is reached during transfer of the workpiece	<p>TC142 (bit 5) = 0: An alarm will not be issued even if the ending position of workpiece pressing is reached during the transfer of the workpiece.</p> <p>TC142 (bit 5) = 1: An alarm will be issued if the ending position of workpiece pressing is reached during the transfer of the workpiece.</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC142 (bit 6)	X-axis retraction position during workpiece transfer as specified in the workpiece transfer unit of the MAZATROL program	<p>TC142 (bit 6) = 0: When the workpiece is transferred as specified in the workpiece transfer unit of the MAZATROL program, the X-axis moves to the machine zero point.</p> <p>TC142 (bit 6) = 1: When the workpiece is transferred as specified in the workpiece transfer unit of the MAZATROL program, the X-axis moves to the third zero point.</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC142 (bit 7)	Upper turret retraction during machining with the lower turret	<p>TC142 (bit 7) = 0: Upper turret moves to the X-axis zero point during machining with the lower turret.</p> <p>TC142 (bit 7) = 1: Upper turret retraction does not occur during machining with the lower turret.</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
TC143	Whether the end tool of the long boring bar can be changed	<p>Whether the end tool of the long boring bar can be changed by ATC (1: Changeable, 0: Not changeable)</p> <p>7 6 5 4 3 2 1 0 (1: Changeable, 0: Not changeable)</p> <ul style="list-style-type: none"> Long boring bar No. 1 Long boring bar No. 2 Long boring bar No. 3 Long boring bar No. 4 Long boring bar No. 5 Long boring bar No. 6 	
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER	Display title	TURNING								
Address	Meaning	Description									
TC144	—	 <ul style="list-style-type: none"> Automatic selection of the relief path for the continuous I.D. machining Movement of the workpiece transfer axis for opposed turret machine <ul style="list-style-type: none"> 0: Z-axis return to zero point, 1: Compliant with TC63/TC64 C-axis clamping during workpiece transfer with C-axis positioning <ul style="list-style-type: none"> 0: Valid 1: Invalid Automatic output of spindle rotation command when turning tool is used in the manual program unit <ul style="list-style-type: none"> 0: Valid 1: Invalid ATC and RETURN of the end unit after single-process operation <ul style="list-style-type: none"> 0: Not executed 1: Executed ATC and RETURN of the end unit for repetition (CONTI. = 1). <ul style="list-style-type: none"> 0: Not executed 1: Executed Relief path G00/G01 selection for edging <ul style="list-style-type: none"> 0: G00 1: G01 Handling when no tools are specified for the manual program unit <ul style="list-style-type: none"> 0: Returning the tool 1: Inheriting the mounted tool 									
	<table border="1" style="margin-bottom: 2px;"> <tr> <td>Program type</td> <td>M</td> </tr> <tr> <td>Conditions</td> <td>After stop of movement</td> </tr> <tr> <td>Unit</td> <td>Bit</td> </tr> <tr> <td>Setting range</td> <td>Binary, eight digits</td> </tr> </table>	Program type	M	Conditions	After stop of movement	Unit	Bit	Setting range	Binary, eight digits		
Program type	M										
Conditions	After stop of movement										
Unit	Bit										
Setting range	Binary, eight digits										

Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC145	—	 <ul style="list-style-type: none"> { OFFSET unit Z-axial inversion during turning 0: Valid 1: Invalid { Handling of the G04 command in the manual program unit 0: Dwell per rev 1: Dwell per min { Operation by cutting-feed override 0 in G31 command mode 0: Normal operation 1: Alarm stop { After C-axis transfer using the workpiece transfer unit 0: Milling mode 1: Turning mode { Selection of a calculation method for tail barrier end reference 0: Spindle edge 1: Z-offset { Output of maximum spindle speed command specified in the common unit 0: At the beginning of each unit 1: Only at the program head (or at the restart position) { Handling when tools different in shape (infeed angle, nose angle and nose R) are designated for the upper and lower turret in balanced cutting 0: Machining continues with the alarm 790 ILLEGAL BALANCE CUT displayed. 1: Machining continues without alarm { Operation of the path inversed with respect to the traveling direction of the tip (only for copy machining) 0: Inversed path ignored 1: Alarm stop 	
		<p><u>Description of TC145 bit 7</u></p> <p>Operation of the path inversed with respect to the traveling direction of the tip</p> <p>0: Inversed path ignored</p> <p>During roughing with the copy-machining unit, if the path is automatically created that is inversed with respect to the traveling direction of the tip, this path will be ignored and the next path executed. In this case, roughing will be executed with a smaller number of cutting operations than the specified number of cutting operations.</p> <p>1: Alarm stop</p> <p>During roughing with the copy-machining unit, if the path is automatically created that is inversed with respect to the traveling direction of the tip, operation will stop with the display of alarm 714 ILLEGAL SEQUENCE DATA to indicate that the path is incorrect. The shape sequence number of the section where the inversed path has been created will be output. Correct the machining program.</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	USER	Display title	TURNING								
Address	Meaning	Description	<p>TC146</p> <p>—</p>  <p>Note : For mill-turning unit or turning-drilling unit in which a milling tool is used, spindle rotates after movement to the relay point, even when TC146 bit 0 = 1.</p> <table border="1" data-bbox="285 1572 714 1718"> <tr> <td>Program type</td><td>M</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>Bit</td></tr> <tr> <td>Setting range</td><td>Binary, eight digits</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Bit	Setting range	Binary, eight digits
Program type	M										
Conditions	Immediate										
Unit	Bit										
Setting range	Binary, eight digits										

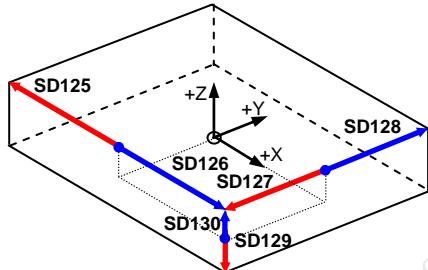
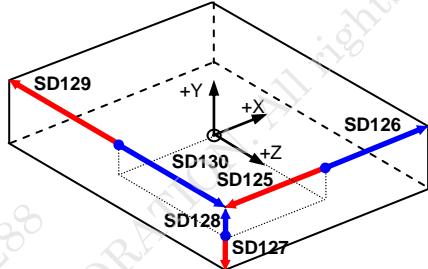
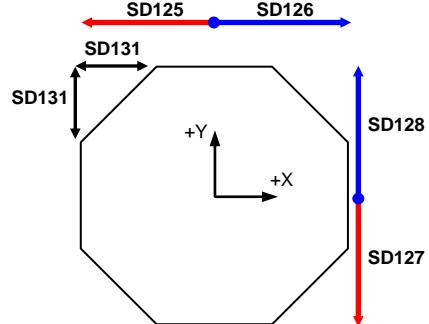
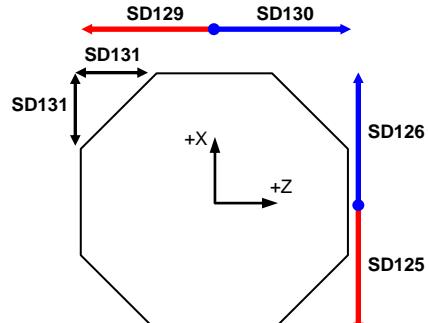
Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC147 (bit 0)	Modal condition about diameter/radius value input for the unit after manual program or subprogram unit	0: The condition of the manual program or subprogram unit is retained. 1: The condition of the unit preceding a manual program or subprogram unit is used.	
	Program type M		
	Conditions Immediate		
	Unit Bit		
	Setting range 0, 1		
TC147 (bit 1)	Z/C offset value to be used in the EIA/ISO subprogram after workpiece transfer in the MAZATROL program	Z/C offset value to be used in the EIA/ISO subprogram after workpiece transfer in the MAZATROL program 0: Z/C offset value set in the workpiece transfer unit 1: Z/C offset value for the MAZATROL program set on the SET UP MANAG. display	
	Program type E		
	Conditions Immediate		
	Unit Bit		
	Setting range 0, 1		
TC147 (bit 4)	Whether or not pressing is to be made selectable for workpiece transfer units of a BAR moving pattern	Whether pressing is to be made selectable for workpiece transfer units of a BAR moving pattern. 0: Not to be made selectable 1: To be made selectable	
	Program type M		
	Conditions Immediate		
	Unit Bit		
	Setting range 0, 1		
TC147 (bit 5)	Tool nose position of an inside-turning grooving tool for machining on the back side	0: Upper right corner of the tip-width section  1: Upper left corner of the tip-width section (compatible with M640T, MT, T NEXUS, and MT Pro) 	
	Program type M		
	Conditions Immediate		
	Unit Bit		
	Setting range 0, 1	* In virtual machining, the machine operates assuming the setting of TC147 bit 5 = 0.	

Classification	USER	Display title	TURNING
Address	Meaning	Description	
TC147 (bit 7)	Z-axis withdrawal after M-code unit execution	<p>Z-axis withdrawal after M-code unit execution when the tool nose is positioned at the edge clearance including the workpiece edge protrusion (See example below.)</p> <p>0: Invalid 1: Valid</p> <p>Example :</p> <p>The tool will be withdrawn after execution of the M-code unit when the tool nose has been positioned in the area between the front clearance point and the workpiece zero point (the hatched section in the figure) using the manual program unit, or when the program is restarted from the M-code unit with the tool nose positioned in that area.</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
	Note : TC147 bit 1 is valid only when F161 bit 7 is set to 1.		
TC148 (bit 0)	Display of an icon for threading position storage	<p>Whether or not an icon is to be displayed as long as the threading position is stored for the re-threading function.</p> <p>0: Not displayed 1: Displayed</p> <p>The icon shown below is displayed:</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
TC149 (bit 5)	Machine operation on tool path check of a program for which workpiece scheme is selected and T. CENTER is specified for X and Y, but Z is left blank in the basic coordinates system unit (WPC).	<p>0: Alarm 613 DATA MISSING IN WPC UNIT occurs. 1: Setting of Z is handled as 0.</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	

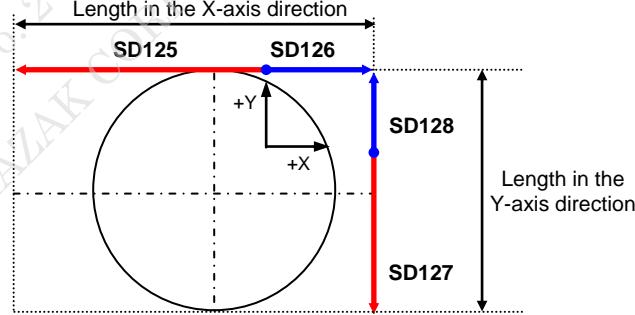
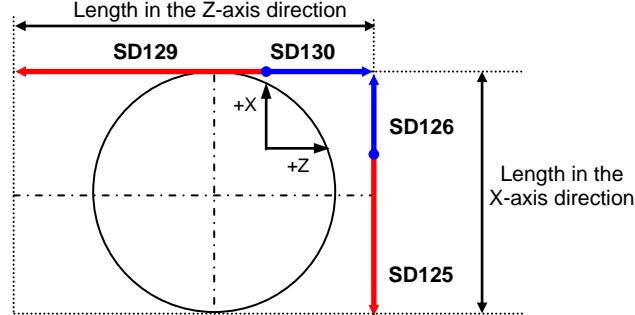
2-3-8 User parameter SOLID (SD)

Classification	USER	Display title	SOLID			
Address	Meaning		Description			
SD49	Machine coordinate system setting		Set the type of coordinate system used for position display. 0: MAZATROL coordinate system 1: Machine coordinate system			
	Program type	M, E				
	Conditions	Immediate				
	Unit	—				
	Setting range	0, 1				
	Table type		Select a table type for 3D setup display. 0: Square table 1: Round table 2: Chuck			
SD50	Program type		Select the axis direction of a round table to be displayed in the 3D SETUP display. 0: X-axis 1: Y-axis 2: Z-axis * This parameter is valid when parameter SD50 is set to 1.			
	Conditions	Immediate				
	Unit	—				
	Setting range	0 to 2				
	Round table axis direction					
	Program type					
SD51	Conditions		Set the default value for the model's movement distance per time that is to be displayed in the movement distance assignment dialog box. Distance of the model movement per time			
	Unit	—				
	Setting range	0 to 2				
	Program type					
	Conditions	Immediate				
	Unit	0.0001 mm/0.00001 in				
SD97	Setting range	0 to 99999999				

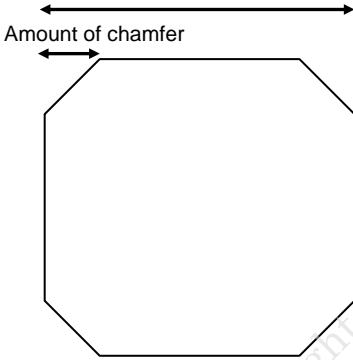
Classification	USER	Display title	SOLID
Address	Meaning	Description	
SD98	Amount of model rotation per time		Set the default value for the amount of model rotation per time that is to be displayed in the amount-of-rotation assignment dialog box.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001°	
	Setting range	0 to 3600000	

Classification	USER	Display title	SOLID
SD125 to SD130	<p>Meaning</p> <p>Table size</p> <p>SD125: Minimum table dimension X SD126: Maximum table dimension X SD127: Minimum table dimension Y SD128: Maximum table dimension Y SD129: Minimum table dimension Z SD130: Maximum table dimension Z</p>	<p>Description</p> <p>Set the table size for 3D setup display.</p> <p>< Rectangular table (not chamfered) ></p> <ul style="list-style-type: none"> Vertical type  <ul style="list-style-type: none"> Horizontal type  <p>< Rectangular table (chamfered) ></p> <ul style="list-style-type: none"> Vertical type  <ul style="list-style-type: none"> Horizontal type 	

Program type	M, E
Conditions	Immediate
Unit	0.0001 mm/0.00001 in
Setting range	0 to ±99999999

Classification	USER	Display title	SOLID
Address		Meaning	Description
SD125 to SD130		<p>Table size</p> <p>SD125: Minimum table dimension X SD126: Maximum table dimension X SD127: Minimum table dimension Y SD128: Maximum table dimension Y SD129: Minimum table dimension Z SD130: Maximum table dimension Z</p>	<p>< Round table ></p> <p>Table upper face direction is determined according to the setting of parameter SD51.</p> <ol style="list-style-type: none"> 1. SD51 = 0: In the length in the Y-axis direction (SD127 + SD128) and that in the Z-axis direction (SD129 + SD130), the value having a smaller absolute value is taken as the diameter of the round table, and the length in the X-axis direction (SD125 + SD126) is taken as the height of the round table. The center of the rectangle (SD127, SD128, SD129, SD130) is taken as the center of the round table. 2. SD51 = 1: In the length in the X-axis direction (SD125 + SD126) and that in the Z-axis direction (SD129 + SD130), the value having a smaller absolute value is taken as the diameter of the round table, and the length in the Y-axis direction (SD127 + SD128) is taken as the height of the round table. The center of the rectangle (SD125, SD126, SD129, SD130) is taken as the center of the round table. 3. SD51 = 2: In the length in the X-axis direction (SD125 + SD126) and that in the Y-axis direction (SD127 + SD128), the value having a smaller absolute value is taken as the diameter of the round table, and the length in the Z-axis direction (SD129 + SD130) is taken as the height of the round table. The center of the rectangle (SD125, SD126, SD127, SD128) is taken as the center of the round table. <ul style="list-style-type: none"> • Vertical type (SD51 = 2)  • Horizontal type (SD51 = 1) 

Classification	USER	Display title	SOLID
Address	Meaning	Description	
SD131	Amount of table chamfer	Set the amount of table chamfer for 3D setup display. <ul style="list-style-type: none"> Top view of the table Length of the shorter side in vertical and horizontal directions	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 in	
	Setting range	0 to ±99999999	



2-3-9 Machine parameter CALL MACRO (J)

Classification	MACHINE		Display title	CALL MACRO						
----------------	---------	--	---------------	------------	--	--	--	--	--	--

G-code macroprogram call

No.1	No.2	No.3	No.4	No.5	No.6	No.7	No.8	No.9	No.10	Unit	Setting range	Program type	Conditions	Description
J1	J5	J9	J13	J17	J21	J25	J29 100009590 (Fixed value)	J33 100009599 (Fixed value)	J37 100009401 (Fixed value)	—	0 to 999999999	M, E	Immediate	Work number of the program to be called
J2	J6	J10	J14	J18	J22	J26	J30 136 (Fixed value)	J34 137 (Fixed value)	J38 130 (Fixed value)	—	0 to 999	M, E	Immediate	The G-code number to be used for program call Note : Not possible to set G-codes whose uses are predefined.
J3	J7	J11	J15	J19	J23	J27	J31 1 (Fixed value)	J35 1 (Fixed value)	J39 2 (Fixed value)	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1
J4	J8	J12	J16	J20	J24	J28	J32	J36	J40	—	—	—	—	Invalid

M-code macroprogram call

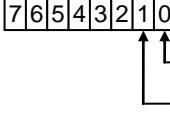
No. 1	No. 2	No. 3	No. 4	No. 5	Unit	Setting range	Program type	Conditions	Description
J41 100000090 (Fixed value)	J45 100000091 (Fixed value)	J49 100000092 (Fixed value)	J53 100000093 (Fixed value)	J57 100000001 (Fixed value)	—	0 to 999999999	M, E	Immediate	Work number of the program to be called
J42 90 (Fixed value)	J46 91 (Fixed value)	J50 92 (Fixed value)	J54 93 (Fixed value)	J58 153 (Fixed value)	—	0 to 9999	M, E	Immediate	The M-code number to be used for program call Note : Not possible to set M-codes whose uses are predefined.
J43 0 (Fixed value)	J47 0 (Fixed value)	J51 0 (Fixed value)	J55 0 (Fixed value)	J59 0 (Fixed value)	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1
J44 0 (Fixed value)	J48 0 (Fixed value)	J52 0 (Fixed value)	J56 0 (Fixed value)	J60 0 (Fixed value)	—	—	—	—	Parameter for system internal setting Setting prohibited

No. 6	No. 7	No. 8	No. 9	No. 10	Unit	Setting range	Program type	Conditions	Description
J61 100000002 (Fixed value)	J65	J69	J73	J77	—	0 to 999999999	M, E	Immediate	Work number of the program to be called
J62 154 (Fixed value)	J66	J70	J74	J78	—	0 to 9999	M, E	Immediate	The M-code number to be used for program call Note : Not possible to set M-codes whose uses are predefined.
J63 0 (Fixed value)	J67	J71	J75	J79	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1
J64 0 (Fixed value)	J68	J72	J76	J80	—	—	—	—	Invalid

Classification	MACHINE	Display title	CALL MACRO
Address	Meaning		Description
J89 to J107	 — Program type — Conditions — Unit — Setting range —		Parameter for system internal setting Setting prohibited
J108	Maximum number of retrials to update the model with the SAFETY SHIELD function Program type — Conditions At power on Unit Times Setting range -99999999 to 99999999		When updating the model with the SAFETY SHIELD function at power-on, if "model update completion check interval (300 ms)* the number of retrials" counting from the start of update is exceeded, it will be regarded as a failure in model update and alarm 1106 SAFETY SHIELD FAILURE (3, ,) will be output. 0 or less : 50 times 1 or above : The setting value (J108) is regarded as the number of retrials.

Classification	MACHINE	Display title	CALL MACRO								
J109 to J112	<p>Barrier setup chuck reference position 1 on the No. 2 spindle side Barrier setup NC tailstock position</p> <table border="1" style="margin-left: 10px;"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>-99999999 to 99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	-99999999 to 99999999	<p>Set the chuck reference position 1 on the No. 2 spindle side or the NC tailstock position with respect to the machine zero point of the upper turret. Use BA79 to BA86 for machines with the No. 2 spindle set as System 2.</p> <ul style="list-style-type: none"> • Setting the chuck reference position 1 on the No. 2 spindle side <ul style="list-style-type: none"> • Setting the NC tailstock position 	
Program type	M, E										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	-99999999 to 99999999										
J113 to J116	<p>Barrier setup chuck reference position 2 on the No. 2 spindle side</p> <table border="1" style="margin-left: 10px;"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>-99999999 to 99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	-99999999 to 99999999	<p>Set the chuck reference position 2 on the No. 2 spindle side with respect to the machine zero point of the upper turret. Use BA79 to BA86 for machines with the No. 2 spindle set as System 2.</p>	
Program type	M, E										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	-99999999 to 99999999										

Classification	MACHINE	Display title	CALL MACRO
Address	Meaning		Description
J119 J120	Barrier-setting/cancelling M-code to be used for barrier teaching		<p>Specify the ID number of the barrier-setting and barrier-cancelling M-codes to be used for barrier teaching function.</p> <p>J119: Barrier-setting M-code to be used for barrier teaching (If a value other than 1 - 999 is set, the value will be regarded as 846.)</p> <p>J120: Barrier-cancelling M-code to be used for barrier teaching (If a value other than 1 - 999 is set, the value will be regarded as 847.)</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	1 to 999	
J131 to J140	Setting the M-code for making the INTELLIGENT SAFETY SHIELD function valid/invalid during automatic operation		<p>Specify the ID number of the M-code for making the interference check function valid/invalid.</p> <p>J131: INTELLIGENT SAFETY SHIELD function invalid J132: Material model cutting invalid J133: Machine interference check invalid J134: Machine interference check valid J135: Material model interference check invalid J136: Material model interference check valid J137: Fixture model interference check invalid J138: Fixture model interference check valid J139: Tool model interference check invalid J140: Tool model interference check valid</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	—	
J141	Selecting whether the M-code registered in parameter J131–J138 to make the INTELLIGENT SAFETY SHIELD function valid/invalid during automatic operation is to be output from the NC to the PLC ladder.		<p>0: Invalid (No output to the PLC ladder) 1: Valid (Output to the PLC ladder)</p> <p>7 6 5 4 3 2 1 0</p> <ul style="list-style-type: none"> M-code registered in J131 M-code registered in J132 M-code registered in J133 M-code registered in J134 M-code registered in J135 M-code registered in J136 M-code registered in J137 M-code registered in J138
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	MACHINE	Display title	CALL MACRO
Address	Meaning		Description
J142	Selecting whether to output the M-code for making the INTELLIGENT SAFETY SHIELD function valid/invalid during automatic operation		<p>Select whether the M-code registered in parameter J139 and J140 to make the INTELLIGENT SAFETY SHIELD function valid/invalid during automatic operation is to be output from the NC to the PLC ladder.</p> <p>0: Invalid (No output to the PLC ladder) 1: Valid (Output to the PLC ladder)</p> 
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
J144 (bit 0)	Tool interference check type		<p>Tool interference check type on the TOOL DATA, TOOL LAYOUT and VISUAL TOOL MANAGEMENT displays</p> <p>0: Magazine-type tool interference check 1: TOOL HIVE-type tool interference check</p>
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	
J144 (bit 2)	Tool life data overwrite during automatic operation		<p>Overwriting of tool life data (maximum allowable wear, life time, use time, life count, use count) when parameter L57 is set to 1 (tool data overwriting during automatic operation valid)</p> <p>0: Possible 1: Impossible</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	MACHINE	Display title	CALL MACRO					
----------------	---------	---------------	------------	--	--	--	--	--

M-code macroprogram call (extended)

No. 11	No .12	No. 13	No. 14	No. 15	Unit	Setting range	Program type	Conditions	Description
J145	J149	J153	J157	J161	—	0 to 99999999	M, E	Immediate	Work number of the program to be called
J146	J150	J154	J158	J162	—	0 to 9999	M, E	Immediate	The M-code number to be used for program call Note : Not possible to set M-codes whose uses are predefined.
J147	J151	J155	J159	J163	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1
J148	J152	J156	J160	J164	—	—	—	—	Invalid

No. 16	No. 17	No. 18	No. 19	No. 20	Unit	Setting range	Program type	Conditions	Description
J165	J169	J173	J177	J181	—	0 to 99999999	M, E	Immediate	Work number of the program to be called
J166	J170	J174	J178	J182	—	0 to 9999	M, E	Immediate	The M-code number to be used for program call Note : Not possible to set M-codes whose uses are predefined.
J167	J171	J175	J179	J183	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1
J168	J172	J176	J180	J184	—	—	—	—	Invalid

Classification	MACHINE	Display title	CALL MACRO				
----------------	---------	---------------	------------	--	--	--	--

G-code macroprogram call (extended)

No.11	No.12	No.13	No.14	No.15	Unit	Setting range	Program type	Conditions	Description
J265	J269	J273	J277	J281	—	0 to 99999999	M, E	Immediate	Work number of the program to be called
J266	J270	J274	J278	J282	—	0 to 9999	M, E	Immediate	The G-code number to be used for program call Note : Not possible to set G-codes whose uses are predefined.
J267	J271	J275	J279	J283	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1
J268	J272	J276	J280	J284	—	—	—	—	Invalid

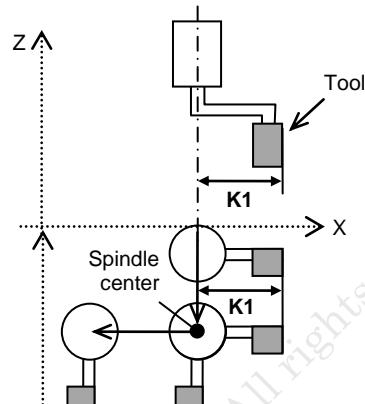
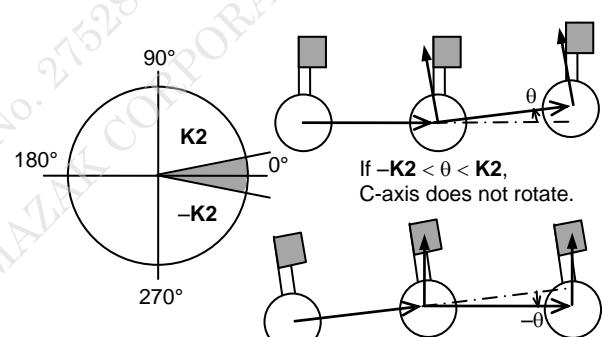
No.16	No.17	No.18	No.19	No.20	Unit	Setting range	Program type	Conditions	Description
J285	J289	J293	J297	J301	—	0 to 99999999	M, E	Immediate	Work number of the program to be called
J286	J290	J294	J298	J302	—	0 to 9999	M, E	Immediate	The G-code number to be used for program call Note : Not possible to set G-codes whose uses are predefined.
J287	J291	J295	J299	J303	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1
J288	J292	J296	J300	J304	—	—	—	—	Invalid

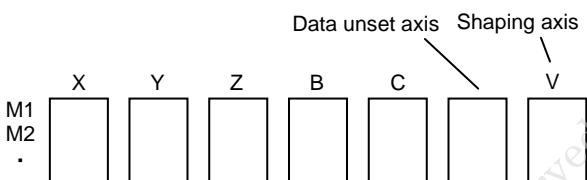
No.21	No.22	No.23	No.24	No.25	Unit	Setting range	Program type	Conditions	Description
J305	J309	J313	J317	J321	—	0 to 99999999	M, E	Immediate	Work number of the program to be called
J306	J310	J314	J318	J322	—	0 to 9999	M, E	Immediate	The G-code number to be used for program call Note : Not possible to set G-codes whose uses are predefined.
J307	J311	J315	J319	J323	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1
J308	J312	J316	J320	J324	—	—	—	—	Invalid

No.26	No.27	No.28	No.29	No.30	Unit	Setting range	Program type	Conditions	Description
J325	J329	J333	J337	J341	—	0 to 99999999	M, E	Immediate	Work number of the program to be called
J326	J330	J334	J338	J342	—	0 to 9999	M, E	Immediate	The G-code number to be used for program call Note : Not possible to set G-codes whose uses are predefined.
J327	J331	J335	J339	J343	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1
J328	J332	J336	J340	J344	—	—	—	—	Invalid

Classification		MACHINE		Display title		CALL MACRO				
No.31	No.32	No.33	No.34	No.35	Unit	Setting range	Program type	Conditions	Description	
J345	J349	J353	J357	J361	—	0 to 999999999	M, E	Immediate	Work number of the program to be called	
J346	J350	J354	J358	J362	—	0 to 9999	M, E	Immediate	The G-code number to be used for program call Note : Not possible to set G-codes whose uses are predefined.	
J347	J351	J355	J359	J363	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1	
J348	J352	J356	J360	J364	—	—	—	—	Invalid	
No.36	No.37	No.38	No.39	No.40	Unit	Setting range	Program type	Conditions	Description	
J365	J369	J373	J377	J381	—	0 to 999999999	M, E	Immediate	Work number of the program to be called	
J366	J370	J374	J378	J382	—	0 to 9999	M, E	Immediate	The G-code number to be used for program call Note : Not possible to set G-codes whose uses are predefined.	
J367	J371	J375	J379	J383	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1	
J368	J372	J376	J380	J384	—	—	—	—	Invalid	
No.41	No.42	No.43	No.44	No.45	Unit	Setting range	Program type	Conditions	Description	
J385	J389	J393	J397	J401	—	0 to 999999999	M, E	Immediate	Work number of the program to be called	
J386	J390	J394	J398	J402	—	0 to 9999	M, E	Immediate	The G-code number to be used for program call Note : Not possible to set G-codes whose uses are predefined.	
J387	J391	J395	J399	J403	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1	
J388	J392	J396	J400	J404	—	—	—	—	Invalid	
No.46	No.47	No.48	No.49	No.50	Unit	Setting range	Program type	Conditions	Description	
J405	J409	J413	J417	J421	—	0 to 999999999	M, E	Immediate	Work number of the program to be called	
J406	J410	J414	J418	J422	—	0 to 9999	M, E	Immediate	The G-code number to be used for program call Note : Not possible to set G-codes whose uses are predefined.	
J407	J411	J415	J419	J423	—	0 to 3	M, E	Immediate	Calling type 0: M98 2: G66 1: G65 3: G66.1	
J408	J412	J416	J420	J424	—	—	—	—	Invalid	

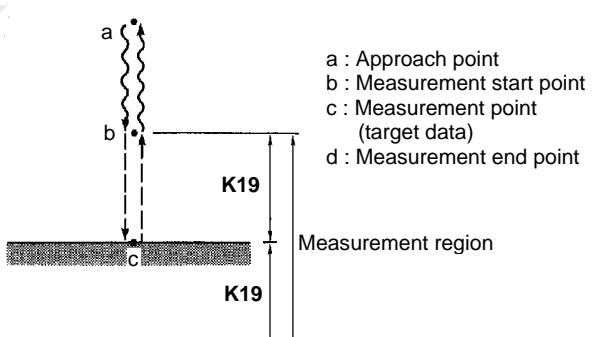
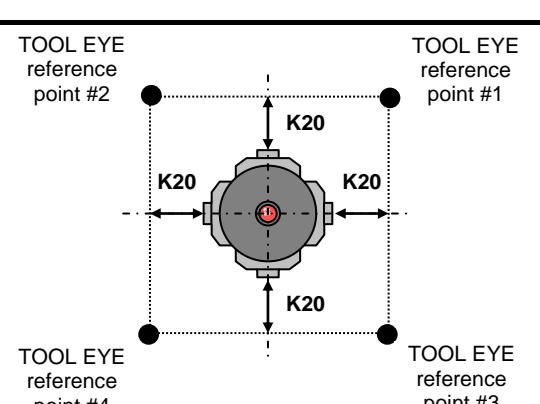
2-3-10 Machine parameter MEASURE (K)

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K1	Rotational radius of the C-axis		<p>Distance from the center of the C-axis (spindle) to the nose of the tool</p> 
	Program type E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to 99999999		
	Minimum rotational angle		<p>The minimum rotational angle of the C-axis at shaping block connections Rotational angle of the C-axis at block connections: θ The C-axis does not rotate if $\theta < K2$.</p> 
	Program type E Conditions Immediate Unit 0.001° Setting range 0 to 90000		

Classification	MACHINE	Display title	MEASURE
K3	Identification number of the shaping control axis	<p>Set identification number of the shaping control axis. Specify the identification number of the shaping control axis as follows according to the value set at parameter K105 bit 6 (handling of the shaping control axis)</p> <p>Example :</p>  <p>When there is an axis for which data is not set on the MACHINE PARAMETER display ("FEED VEL." (M), "TIME CONST." (N), etc.), where the axis data is set, as shown below, set the identification number of the shaping axis as follows:</p> <p>If the value set at parameter K105 bit 6 is: "0", set "6". Counting of the axis number starts with "1" and the axis for which the data is not set is not included when counting the axis number. "1", set "7". Counting of the axis number starts with "1" and the axis for which the data is not set is included when counting the axis number.</p> <p>Note : Set the type of the axis which has been set on this parameter to the rotational axis (M17 bit 4 = 1).</p>	
K7	Unbalanced axis	Specify the axis that moves vertically 1 : The X-axis acts as the unbalanced axis. 2 : The Y-axis acts as the unbalanced axis. 4 : The Z-axis acts as the unbalanced axis.	
K10	—	Fixed value	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	1, 2, 4	

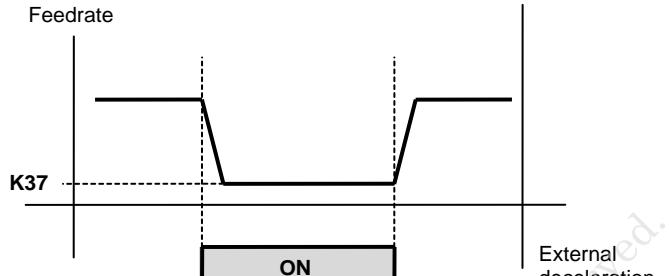
Classification	MACHINE	Display title	MEASURE																																																			
Address	Meaning	Description																																																				
K11	Selection of language to be displayed		Set this parameter to change the display language.																																																			
			<table border="1"> <thead> <tr> <th>Setting</th><th>Language</th><th>Setting</th><th>Language</th></tr> </thead> <tbody> <tr><td>0</td><td>English</td><td>11</td><td>Korean</td></tr> <tr><td>1</td><td>Japanese</td><td>12</td><td>Portuguese</td></tr> <tr><td>2</td><td>German</td><td>13</td><td>Danish</td></tr> <tr><td>3</td><td>French</td><td>14</td><td>Czech</td></tr> <tr><td>4</td><td>Italian</td><td>15</td><td>Turkish</td></tr> <tr><td>5</td><td>Spanish</td><td>16</td><td>Polish</td></tr> <tr><td>6</td><td>Norwegian</td><td>17</td><td>Romanian</td></tr> <tr><td>7</td><td>Swedish</td><td>18</td><td>Hungarian</td></tr> <tr><td>8</td><td>Finnish</td><td>19</td><td>Russian</td></tr> <tr><td>9</td><td>Chinese (traditional character)</td><td>20</td><td>Slovak</td></tr> <tr><td>10</td><td>Dutch</td><td>21</td><td>Chinese (simplified character)</td></tr> <tr><td></td><td></td><td>25</td><td>Bulgarian</td></tr> </tbody> </table>	Setting	Language	Setting	Language	0	English	11	Korean	1	Japanese	12	Portuguese	2	German	13	Danish	3	French	14	Czech	4	Italian	15	Turkish	5	Spanish	16	Polish	6	Norwegian	17	Romanian	7	Swedish	18	Hungarian	8	Finnish	19	Russian	9	Chinese (traditional character)	20	Slovak	10	Dutch	21	Chinese (simplified character)			25
Setting	Language	Setting	Language																																																			
0	English	11	Korean																																																			
1	Japanese	12	Portuguese																																																			
2	German	13	Danish																																																			
3	French	14	Czech																																																			
4	Italian	15	Turkish																																																			
5	Spanish	16	Polish																																																			
6	Norwegian	17	Romanian																																																			
7	Swedish	18	Hungarian																																																			
8	Finnish	19	Russian																																																			
9	Chinese (traditional character)	20	Slovak																																																			
10	Dutch	21	Chinese (simplified character)																																																			
		25	Bulgarian																																																			
Program type	—																																																					
Conditions	Immediate																																																					
Unit	—																																																					
Setting range	0 to 21, 25																																																					
K12	—		Fixed value (0)																																																			
	Program type	—																																																				
	Conditions	—																																																				
	Unit	—																																																				
	Setting range	—																																																				

Classification	MACHINE	Display title	MEASURE
K13	Measurement skip feed rate (X-axis, Z-axis)		<p>1. Workpiece measurement</p> <p>Sensor path fa K13 Workpiece Measurement stroke (K19)</p> <p>fa: Measurement approach speed (K14)</p> <p>NM211-00233</p>
K14	Measurement approach speed (X-axis, Z-axis)		<p>2. Tool tip measurement</p> <p>Tool path K13 Sensor Measurement stroke (K20)</p> <p>(⇒K19, K20)</p> <p>NM211-00234</p>
K15	Measurement skip speed (C-axis)		<p>Sensor path [1] [2] [3] [4] Workpiece</p> <p>[1], [3] : Rapid feed [2] : Measurement approach speed (K16) [4] : K15</p> <p>NM211-00235</p>
K16	Measurement approach speed (C-axis)		
	Program type	M	
	Conditions	Immediate	
	Unit	°/min	
	Setting range	0 to 65535	

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K17	Specification of measuring tolerance (lower limit)		<p>1. Tool compensation will be made in cases below.</p> $\frac{\alpha_1 - \alpha_2}{100} \times K18 \geq \text{Compensation data} \geq \frac{\alpha_1 - \alpha_2}{100} \times K17$ $-\frac{\alpha_1 - \alpha_2}{100} \times K17 \geq \text{Compensation data} \geq -\frac{\alpha_1 - \alpha_2}{100} \times K18$ <p>2. Tool compensation will not be made in cases below.</p> $\frac{\alpha_1 - \alpha_2}{100} \times K17 > \text{Compensation data} > -\frac{\alpha_1 - \alpha_2}{100} \times K17$ <p>3. Alarm will be caused in cases below.</p> $\text{Compensation data} > \frac{\alpha_1 - \alpha_2}{100} \times K18$ $-\frac{\alpha_1 - \alpha_2}{100} \times K18 > \text{Compensation data}$ <p>where α_1 = Tolerance upper limit α_2 = Tolerance lower limit (Specified in program)</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
K18	Specification of measuring tolerance (upper limit)		<p>Note :</p> <ol style="list-style-type: none"> Up to 100 % can be set. Offset judgement occurs only when L106 bit 6 ist set to 1.
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
K19	Measurement stroke for workpiece measurement		 <p>a : Approach point b : Measurement start point c : Measurement point (target data) d : Measurement end point</p> <p>K19</p> <p>Measurement region</p> <p>NM211-00259</p>
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 65535	
K20	Measurement stroke for tool nose measurement		 <p>TOOL EYE reference point #2</p> <p>K20</p> <p>K20</p> <p>K20</p> <p>K20</p> <p>TOOL EYE reference point #4</p> <p>TOOL EYE reference point #3</p>
	Program type	M	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 65535	

Classification	MACHINE	Display title	MEASURE								
Address	Meaning	Description									
K21	<p>Coefficient to determine rotation angle when retrying measurement C reference face</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>%</td></tr> <tr><td>Setting range</td><td>1 to 1000</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	%	Setting range	1 to 1000	<p>During approach operation to measurement start point, if touch sensor is actuated, the C-axis will rotate by angle determined by the measurement object shape angle and setting value K21.</p> <p>Example : CW programmed as measurement direction</p> <p>α : Measurement object shape angle β : $\alpha \times K21/100$</p> <p>C-axis rotation by angle β in this direction</p> <p>NM211-00261</p>	
Program type	M										
Conditions	Immediate										
Unit	%										
Setting range	1 to 1000										
K22	<p>Measurement retry frequency when retrying reference face C measurement</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Times</td></tr> <tr><td>Setting range</td><td>0 to 255</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Times	Setting range	0 to 255	<p>When touch sensor is actuated before reaching target point in C offset measurement, set retry frequency.</p> <p>When K22 is set to N (N = 0 to 255), measurement alarm will be indicated when touch sensor is actuated before reaching measurement target point in (N+1)th retry operation.</p>	
Program type	M										
Conditions	Immediate										
Unit	Times										
Setting range	0 to 255										
K23	<p>Retry frequency for workpiece measurement</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Times</td></tr> <tr><td>Setting range</td><td>0 to 99999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	Times	Setting range	0 to 99999	<p>Specify the number of times the workpiece measurement is to be retried if a measurement error occurs.</p>	
Program type	M										
Conditions	Immediate										
Unit	Times										
Setting range	0 to 99999										
K24 to K28	—	<p>Parameter for system internal setting</p> <p>Setting prohibited</p>									
	<table border="1"> <tr><td>Program type</td><td>—</td></tr> <tr><td>Conditions</td><td>—</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>—</td></tr> </table>	Program type	—	Conditions	—	Unit	—	Setting range	—		
Program type	—										
Conditions	—										
Unit	—										
Setting range	—										

Classification	MACHINE	Display title	MEASURE
Address	Meaning	Description	
K29	Simultaneous control: Delay counter for automatic correction of synchronizing errors		Delays master/slave axis position matching during automatic error correction (parameter K103 bit 1 = 0).
	Program type	M, E	Note : The delay time will be 3 seconds if 0 is set.
	Conditions	Immediate	
	Unit	1.7 ms	
	Setting range	0 to 99999	
K30 to K33	K30 Approach speed for laser tool length measurement K31 Approach speed for laser tool diameter measurement K32 Pre-measuring speed for laser tool length measurement K33 Pre-measuring speed for laser tool diameter measurement		Specify the approach speed and pre-measuring speed for laser tool length (diameter) measurement.
	Program type	M, E	
	Conditions	At power on	
	Unit	mm/min / 0.1 in/min	
	Setting range	±99999999	
K34 K35	K34 Pre-measuring spindle speed for laser tool length measurement K35 Pre-measuring spindle speed for laser tool diameter measurement		Specify the pre-measuring spindle speed for laser tool length (diameter) measurement.
	Program type	M, E	
	Conditions	At power on	
	Unit	min ⁻¹	
	Setting range	0 to 65535	
K36	—		Parameter for system internal setting Setting prohibited
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE								
Address	Meaning	Description									
K37	<p>External deceleration speed</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>mm/min</td></tr> <tr><td>Setting range</td><td>0 to 120000</td></tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	mm/min	Setting range	0 to 120000	<p>The upper-limit value of the feed rates available while the external deceleration signal is ON</p>  <p>MPL508</p>	
Program type	M, E										
Conditions	After stop of movement										
Unit	mm/min										
Setting range	0 to 120000										
K38	<p>Work number called during S-code macroprogram appointment</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 999999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0 to 999999999	<p>The work number of the macroprogram to be called during S-code macroprogram appointment (Programming of "S0000;" causes execution of the macroprogram whose work number is set using this parameter.)</p> <p>Note : This parameter is valid only when bit 2 of parameter K105 is 1.</p>	
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0 to 999999999										
K39	<p>Work number called during T-code macroprogram appointment</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 999999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0 to 999999999	<p>The work number of the macroprogram to be called during T-code macroprogram appointment (Programming of "T0000;" causes execution of the macroprogram whose work number is set using this parameter.)</p> <p>Note : This parameter is valid only when bit 3 of parameter K105 is 1.</p>	
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0 to 999999999										
K40	<p>Work number called during second auxiliary function macroprogram appointment</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 999999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0 to 999999999	<p>The work number of the macroprogram to be called during macroprogram appointment using the second auxiliary function</p> <p>Note :</p> <ol style="list-style-type: none"> This parameter is valid only when bit 4 of parameter K105 is 1. See the description of parameter K56 for details of the addresses available with the second auxiliary function. 	
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0 to 999999999										

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K41	G31 skipping speed		The feed rate during axis movement by G31 (skip function) If the same block as that of G31 contains an F command, then that feed rate becomes valid.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	mm/min	
	Setting range	0 to 120000	
K42	G31.1 skipping speed		The feed rate during axis movement by G31.1 (multi-step skip function) If the same block as that of G31.1 contains an F command, then that feed rate becomes valid.
	Program type	E	
	Conditions	After stop of movement	
	Unit	mm/min	
	Setting range	0 to 120000	
K43	G31.2 skipping speed		The feed rate during axis movement by G31.2 (multi-step skip function) If the same block as that of G31.2 contains an F command, then that feed rate becomes valid.
	Program type	E	
	Conditions	After stop of movement	
	Unit	mm/min	
	Setting range	0 to 120000	
K44	G31.3 skipping speed		The feed rate during axis movement by G31.3 (multi-step skip function) If the same block as that of G31.3 contains an F command, then that feed rate becomes valid.
	Program type	E	
	Conditions	After stop of movement	
	Unit	mm/min	
	Setting range	0 to 120000	

Classification	MACHINE	Display title	MEASURE
Address	Meaning	Description	
K45	G31.4 skipping speed Program type E Conditions After stop of movement Unit mm/min Setting range 0 to 120000	The feed rate during axis movement by G31.4 (skip function) If the same block as that of G31.4 contains an F command, then that feed rate becomes valid.	
K46	Excessive pressing error spread (Amount of drooping) Program type M, E Conditions Immediate Unit 0.0001 mm Setting range 0 to 32767	Set the excessive pressing error spread (the amount of drooping).	
K48	Spindle FB alarm detection method Program type M, E Conditions Immediate Unit — Setting range 0 to 2	0: Monitoring based on acceleration 1: Monitoring based on speed 2: Ignoring alarms	
K49	First number of the standby M-codes Program type M, E Conditions After stop of movement Unit — Setting range 31 to 1000	Set the first number of the M-codes to be used for the M-code standby function. Example : To use M950 to M995 as the standby M-codes set "950" in K49 and "46" in K50.	

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K50	Total number of the standby M-codes		<p>Set the total number of M-codes to be used for the M-code standby function.</p> <p>Example : To use M950 to M995 as the standby M-codes set "950" in K49 and "46" in K50.</p> <p>Note : If "0" is set, the M-code standby function will be invalid.</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 1000	
K51	M-code during workpiece measurement retry operation		<p>Set the M-code to be output if workpiece measurement results in an error and is to be retried.</p> <p>Note : If "0" is set, this function will be invalid.</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 1000	
K52	—		<p>Parameter for system internal setting</p> <p>Setting prohibited</p>
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE																																																																																				
K53	Address	Meaning	<p>Description</p> <p>Set the type of vocal output language.</p> <p>MATRIX</p> <table border="1"> <thead> <tr> <th>Setting</th> <th>Language</th> <th>Setting</th> <th>Language</th> </tr> </thead> <tbody> <tr><td>0</td><td>English</td><td>8</td><td>Finnish</td></tr> <tr><td>1</td><td>Japanese</td><td>9</td><td>Chinese (traditional character)</td></tr> <tr><td>2</td><td>German</td><td>10</td><td>—</td></tr> <tr><td>3</td><td>French</td><td>11</td><td>Korean</td></tr> <tr><td>4</td><td>Italian</td><td>12</td><td>Portuguese</td></tr> <tr><td>5</td><td>Spanish</td><td>13 to 20</td><td>—</td></tr> <tr><td>6</td><td>—</td><td>21</td><td>Chinese (simplified character)</td></tr> <tr><td>7</td><td>—</td><td></td><td></td></tr> </tbody> </table> <p>MATRIX 2</p> <table border="1"> <thead> <tr> <th>Setting</th> <th>Language</th> <th>Setting</th> <th>Language</th> </tr> </thead> <tbody> <tr><td>0</td><td>English</td><td>11</td><td>Korean</td></tr> <tr><td>1</td><td>Japanese</td><td>12</td><td>Portuguese</td></tr> <tr><td>2</td><td>German</td><td>13</td><td>Danish</td></tr> <tr><td>3</td><td>French</td><td>14</td><td>Czech</td></tr> <tr><td>4</td><td>Italian</td><td>15</td><td>Turkish</td></tr> <tr><td>5</td><td>Spanish</td><td>16</td><td>Polish</td></tr> <tr><td>6</td><td>Norwegian</td><td>17</td><td>Rumanian</td></tr> <tr><td>7</td><td>Swedish</td><td>18</td><td>Hungarian</td></tr> <tr><td>8</td><td>Finnish</td><td>19</td><td>Russian</td></tr> <tr><td>9</td><td>Chinese (traditional character)</td><td>20</td><td>Slovakian</td></tr> <tr><td>10</td><td>Dutch</td><td>21</td><td>Chinese (simplified character)</td></tr> </tbody> </table> <p>Note : Output will be in English (0) if the selected value is for a language not supported.</p> <p>Program type M, E Conditions Immediate Unit — Setting range 0 to 21</p>	Setting	Language	Setting	Language	0	English	8	Finnish	1	Japanese	9	Chinese (traditional character)	2	German	10	—	3	French	11	Korean	4	Italian	12	Portuguese	5	Spanish	13 to 20	—	6	—	21	Chinese (simplified character)	7	—			Setting	Language	Setting	Language	0	English	11	Korean	1	Japanese	12	Portuguese	2	German	13	Danish	3	French	14	Czech	4	Italian	15	Turkish	5	Spanish	16	Polish	6	Norwegian	17	Rumanian	7	Swedish	18	Hungarian	8	Finnish	19	Russian	9	Chinese (traditional character)	20	Slovakian	10	Dutch	21	Chinese (simplified character)
Setting	Language	Setting	Language																																																																																				
0	English	8	Finnish																																																																																				
1	Japanese	9	Chinese (traditional character)																																																																																				
2	German	10	—																																																																																				
3	French	11	Korean																																																																																				
4	Italian	12	Portuguese																																																																																				
5	Spanish	13 to 20	—																																																																																				
6	—	21	Chinese (simplified character)																																																																																				
7	—																																																																																						
Setting	Language	Setting	Language																																																																																				
0	English	11	Korean																																																																																				
1	Japanese	12	Portuguese																																																																																				
2	German	13	Danish																																																																																				
3	French	14	Czech																																																																																				
4	Italian	15	Turkish																																																																																				
5	Spanish	16	Polish																																																																																				
6	Norwegian	17	Rumanian																																																																																				
7	Swedish	18	Hungarian																																																																																				
8	Finnish	19	Russian																																																																																				
9	Chinese (traditional character)	20	Slovakian																																																																																				
10	Dutch	21	Chinese (simplified character)																																																																																				
K54	Address	Sound level (vocal output)	<p>Description</p> <p>Set the vocal output sound level.</p> <p>Note : Entry of "0" means no sound level.</p> <p>Program type M, E Conditions Immediate Unit — Setting range 0 to 100</p>																																																																																				

Classification	MACHINE	Display title	MEASURE									
Address	Meaning	Description										
K55	Warning reference value (vocal output)		A vocal warning will be output if the value of the spindle load meter exceeds the percentage value set in K55 .									
	Program type	M, E										
	Conditions	Immediate										
	Unit	%										
	Setting range	0 to 200										
K56	Name of second auxiliary function		Select the address name of the second auxiliary function from among the following three types:									
			<table border="1"> <thead> <tr> <th>Address name</th><th>Setting (HEX)</th></tr> </thead> <tbody> <tr> <td>Invalid</td><td>0</td></tr> <tr> <td>A</td><td>41</td></tr> <tr> <td>B</td><td>42</td></tr> <tr> <td>C</td><td>43</td></tr> </tbody> </table>	Address name	Setting (HEX)	Invalid	0	A	41	B	42	C
Address name	Setting (HEX)											
Invalid	0											
A	41											
B	42											
C	43											
Program type	E											
Conditions	At power on											
Unit	—											
K57	Type of S-code macroprogram appointment call		This parameter is used during S-code macroprogram appointment to select the method of calling the macroprogram whose work number has been set using the K38 parameter.									
			<table border="1"> <thead> <tr> <th>Setting</th><th>Calling method</th></tr> </thead> <tbody> <tr> <td>0</td><td>M98 P□□□□</td></tr> <tr> <td>1</td><td>G65 P□□□□</td></tr> <tr> <td>2</td><td>G66 P□□□□</td></tr> <tr> <td>3</td><td>G66.1 P□□□□</td></tr> </tbody> </table>	Setting	Calling method	0	M98 P□□□□	1	G65 P□□□□	2	G66 P□□□□	3
Setting	Calling method											
0	M98 P□□□□											
1	G65 P□□□□											
2	G66 P□□□□											
3	G66.1 P□□□□											
Program type	M, E											
Conditions	Immediate											
Unit	—											
K58	Type of T-code macroprogram appointment call		This parameter is used during T-code macroprogram appointment to select the method of calling the macroporgram whose work number has been set using the K39 parameter.									
			<table border="1"> <thead> <tr> <th>Setting</th><th>Calling method</th></tr> </thead> <tbody> <tr> <td>0</td><td>M98 P□□□□</td></tr> <tr> <td>1</td><td>G65 P□□□□</td></tr> <tr> <td>2</td><td>G66 P□□□□</td></tr> <tr> <td>3</td><td>G66.1 P□□□□</td></tr> </tbody> </table>	Setting	Calling method	0	M98 P□□□□	1	G65 P□□□□	2	G66 P□□□□	3
Setting	Calling method											
0	M98 P□□□□											
1	G65 P□□□□											
2	G66 P□□□□											
3	G66.1 P□□□□											
Program type	M, E											
Conditions	Immediate											
Unit	—											
	Setting range	0 to 3										

Classification	MACHINE	Display title	MEASURE
Address	Meaning	Description	
K59	Type of second auxiliary function macroprogram appointment call		This parameter is used during the second auxiliary function macroprogram appointment to select the method of calling the macropogram whose work number has been set using the K40 parameter.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 3	
K60	—		Fixed value (4)
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K61	—		Fixed value (1)
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K62	—		Fixed value (1)
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K63	—	Fixed value (1)	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K64	—	Fixed value (2)	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K65	—	Fixed value (1)	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K66	—	Fixed value (1)	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K67	—		Fixed value (1)
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K68 (bit 0)	Coordinate system in which the amount of tool offset becomes valid (for five-face machining by angular tools)		Coordinate system with reference to which the offset amounts are to be used in tool length offset for the selected machining surface. (For machines of five-surface machining by angular tools) 0: Machine coordinate system 1: Workpiece coordinate system
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
K68 (bit 4)	—		Fixed value (0)
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K68 (bit 5)	Override/dryrun speed valid/invalid when the G31 command is issued		0 : Invalid 1 : Valid
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K68 (bit 7)	Whether or not all the barrier cancel data in the program is to be cleared by executing the MMS unit in the MAZATROL program		0: Invalid 1: Valid
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
K69 (bit 0)	G31.1 skip condition SKIP-2		In the execution of the G31.1 command, whether or not SKIP-2 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K69 (bit 1)	G31.1 skip condition SKIP-10		In the execution of the G31.1 command, whether or not SKIP-10 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K69 (bit 2)	G31.1 skip condition SKIP-3		In the execution of the G31.1 command, whether or not SKIP-3 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	

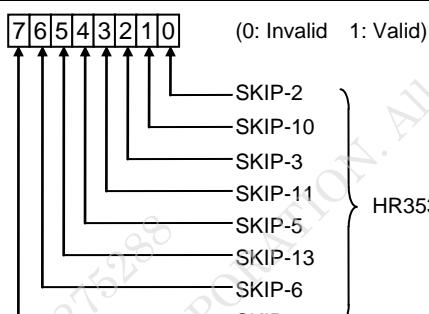
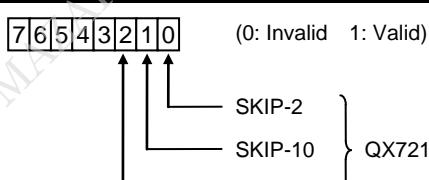
Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K69 (bit 3)	G31.1 skip condition SKIP-11	Program type E Conditions After stop of movement Unit Bit Setting range 0, 1	In the execution of the G31.1 command, whether or not SKIP-11 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
K69 (bit 4)	G31.1 skip condition SKIP-5	Program type E Conditions After stop of movement Unit Bit Setting range 0, 1	In the execution of the G31.1 command, whether or not SKIP-5 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
K69 (bit 5)	G31.1 skip condition SKIP-13	Program type E Conditions After stop of movement Unit Bit Setting range 0, 1	In the execution of the G31.1 command, whether or not SKIP-13 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
K69 (bit 6)	G31.1 skip condition SKIP-6	Program type E Conditions After stop of movement Unit Bit Setting range 0, 1	In the execution of the G31.1 command, whether or not SKIP-6 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K69 (bit 7)	G31.1 skip condition SKIP-14		In the execution of the G31.1 command, whether or not SKIP-14 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K70 (bit 0)	G31.2 skip condition SKIP-2		In the execution of the G31.2 command, whether or not SKIP-2 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K70 (bit 1)	G31.2 skip condition SKIP-10		In the execution of the G31.2 command, whether or not SKIP-10 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K70 (bit 2)	G31.2 skip condition SKIP-3		In the execution of the G31.2 command, whether or not SKIP-3 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K70 (bit 3)	G31.2 skip condition SKIP-11	Program type E Conditions After stop of movement Unit Bit Setting range 0, 1	In the execution of the G31.2 command, whether or not SKIP-11 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
K70 (bit 4)	G31.2 skip condition SKIP-5	Program type E Conditions After stop of movement Unit Bit Setting range 0, 1	In the execution of the G31.2 command, whether or not SKIP-5 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
K70 (bit 5)	G31.2 skip condition SKIP-13	Program type E Conditions After stop of movement Unit Bit Setting range 0, 1	In the execution of the G31.2 command, whether or not SKIP-13 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
K70 (bit 6)	G31.2 skip condition SKIP-6	Program type E Conditions After stop of movement Unit Bit Setting range 0, 1	In the execution of the G31.2 command, whether or not SKIP-6 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K70 (bit 7)	G31.2 skip condition SKIP-14		In the execution of the G31.2 command, whether or not SKIP-14 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K71 (bit 0)	G31.3 skip condition SKIP-2		In the execution of the G31.3 command, whether or not SKIP-2 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K71 (bit 1)	G31.3 skip condition SKIP-10		In the execution of the G31.3 command, whether or not SKIP-10 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K71 (bit 2)	G31.3 skip condition SKIP-3		In the execution of the G31.3 command, whether or not SKIP-3 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	

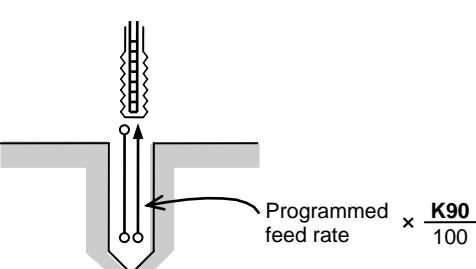
Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K71 (bit 3)	G31.3 skip condition SKIP-11		In the execution of the G31.3 command, whether or not SKIP-11 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K71 (bit 4)	G31.3 skip condition SKIP-5		In the execution of the G31.3 command, whether or not SKIP-5 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K71 (bit 5)	G31.3 skip condition SKIP-13		In the execution of the G31.3 command, whether or not SKIP-13 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K71 (bit 6)	G31.3 skip condition SKIP-6		In the execution of the G31.3 command, whether or not SKIP-6 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	

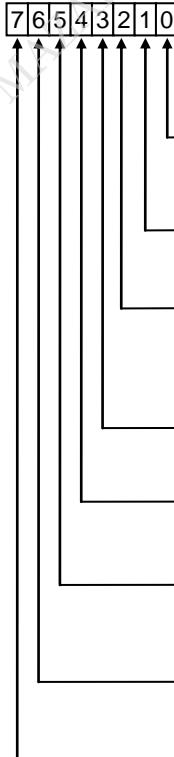
Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K71 (bit 7)	G31.3 skip condition SKIP-14		In the execution of the G31.3 command, whether or not SKIP-14 is used as the skip signal: 0: Not used as the skip signal 1: Used as the skip signal
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K72	G31.4 skip conditions		 <p>Select the skip signal for the G31.4 command.</p>
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
K73	G4 skip conditions		 <p>Select the skip signal for G4 command.</p>
	Program type	E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
K74	Emergency stop contactor cutoff time (Safety supervisory function)		<p>Set the desired time from an emergency stop to the start of cutting off the contactor of the main power to the driving section during the execution of a safety supervisory function when all-axis zero-speed confirmation is impossible.</p> <p>A contactor cutoff signal will be immediately output if all-axis zero-speed confirmation is executable earlier than the set time.</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	s	
	Setting range	0 to 60	

Classification	MACHINE	Display title	MEASURE								
Address	Meaning	Description									
K75	<p>Contactor control output device 1 (Safety supervisory function)</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>&0 to &7F</td></tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	—	Setting range	&0 to &7F	<p>Specify a remote I/O device that is to output a contactor activating/deactivating signal when the safety supervisory function is executed.</p> <p>Note : The signal will not be output if “&0” is specified.</p>	
Program type	M, E										
Conditions	After stop of movement										
Unit	—										
Setting range	&0 to &7F										
K76	<p>Contactor control output device 2 (Safety supervisory function)</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>&0 to &7F</td></tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	—	Setting range	&0 to &7F	<p>Specify another remote I/O device that is to output the contactor activating/deactivating signal when the safety supervisory function is executed.</p> <p>Note : The signal will not be output if “&0” is specified.</p>	
Program type	M, E										
Conditions	After stop of movement										
Unit	—										
Setting range	&0 to &7F										
K77	<p>Door switch input device (Safety supervisory function)</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>&0 to &7F</td></tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	—	Setting range	&0 to &7F	<p>Enter the device number of the remote I/O device to be activated to input a door open/closed status signal. The device of the entered serial device number will be reserved according to the particular setting of K78 (the parameter for setting the number of door switches).</p> <p>Note : Door switch input will be invalid if “&0” is entered.</p>	
Program type	M, E										
Conditions	After stop of movement										
Unit	—										
Setting range	&0 to &7F										
K78	<p>Number of door switches (Safety supervisory function)</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 16</td></tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	—	Setting range	0 to 16	Set the number of doors for which the door open/closed status signal is to be output.	
Program type	M, E										
Conditions	After stop of movement										
Unit	—										
Setting range	0 to 16										

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K79	Supervisory speed filtering time during servo-off (Safety supervisory function)		Set the filtering time for the speed that is to be monitored in a servo-off status during safety speed monitoring. Note : Input of "0" means 200 ms.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	1.777 ms	
	Setting range	0 to 1000	
K80	Warning reference value (Vocal output)		A vocal warning will be output when the value of the load meter for servo axis control reaches or exceeds the percentage value set in K80.
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 200	
K81	Timer for machining management		Fixed value
	Program type	M	
	Conditions	After stop of movement	
	Unit	ms	
	Setting range	0 to 32767	
K82	—		Fixed value
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

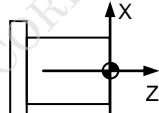
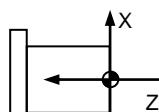
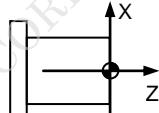
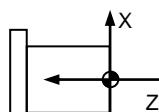
Classification	MACHINE	Display title	MEASURE
Address	Meaning	Description	
K83	Maximum amount of compensation for workpiece thermal expansion	<p>Specify the maximum amount of compensation for workpiece thermal expansion. In order to prevent eventual collision, the maximum amount should not exceed the distance between software limit and mechanical end on any axis concerned.</p> <p>Note :</p> <ol style="list-style-type: none"> 1. If 0 is set here, compensation will not be made for workpiece thermal expansion. 2. This parameter is valid for machining centers with the capability of compensating for workpiece thermal expansion. 	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.00005 mm	
	Setting range	0 to 32767	
K84	Maximum allowable amount of compensation for workpiece thermal expansion in each interpolating cycle	<p>Specify the maximum amount of compensation for workpiece thermal expansion in each interpolating cycle.</p> <p>If the compensation amount computed exceeds the maximum value (as set with K84), compensation will be executed by dividing the interpolation motion into multiple sections.</p> <p>Note :</p> <ol style="list-style-type: none"> 1. If 0 is set here, the amount is set to 0.001 mm. 2. This parameter is valid for machining centers with the capability of compensating for workpiece thermal expansion. 	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.00005 mm	
	Setting range	0 to 32767	
K85	Special linear acceleration/deceleration time constant for threading	<p>If the setting of K85 is from 1 to 300 ms, this setting will be used as the linear acceleration/deceleration time constant for the G32 threading block.</p> <p>If the setting is outside the valid range, however, the normal linear acceleration/deceleration time constant for G01 will be used.</p>	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	ms	
	Setting range	0 to 32767	
K87 to K89	—	Parameter for system internal setting	Setting prohibited
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K90	Return override during synchronous tapping		The overriding value for return from the hole bottom during a synchronous tapping cycle  Note : This parameter is valid only when bit 6 of F94 is 1. MPL509
	Program type	E	
	Conditions	After stop of movement	
	Unit	%	
	Setting range	0 to 999	
K91	Alternative M-code for M96		Specify an alternative M-code for M96 when user macro interruption is valid.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 127	
K92	Alternative M-code for M97		Specify an alternative M-code for M97 when user macro interruption is valid.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 127	
K93	—		Fixed value (2)
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K94 (bit 0)	Selection of whether or not the menu function is to be made invalid after operation has been stopped by [ATC STOP] Program type M, E Conditions Immediate Unit Bit Setting range 0, 1		If the menu function does not need to be made invalid to restart the automatic operation that has been stopped by the [ATC STOP] menu function, set this parameter to "1" (only for applicable machines). 0: [ATC STOP] is to be made invalid 1: [ATC STOP] is not to be made invalid
K94 (bit 3)	W-axis command code selection for MDI mode Program type M, E Conditions Immediate Unit Bit Setting range 0, 1		Command code to be displayed in the MDI window in response to the selection of the [W SUB] MDI menu item 0: G110B2 1: G110Z[B]2
K95	— Program type M, E Conditions At power on Unit Bit Setting range Binary, eight digits		 <ul style="list-style-type: none"> Select whether the difference in angle between the upper and lower turrets is to be incorporated into the current C-axis position display of HD2. 0: Yes (Incorporated) 1: No (Not incorporated) Fixed value (0) Tool position compensation during T-command execution 0: Not performed 1: Performed Coordinate system update during handle pulse interrupt 0: Not performed 1: Performed Fixed value (0) Acceleration/deceleration time constant for handle pulse feed 0: Time constant for cutting feed 1: No time constant Software limits for G30 execution 0: Invalid 1: Valid In-position check 0: Invalid 1: Valid

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K96	—		<p>G0 command in-position check 0: Check 1: Non-check</p> <p>Timing of manual free feed finish signal 0: Smoothing 1: Distribution finish (equivalent to DEN)</p> <p>Fixed value (0)</p> <p>Axis/Cutting interlock alarm display 0: Valid 1: Invalid</p> <p>Suppression of lost motion in modes other than the G1 command mode 0: Valid 1: Invalid</p> <p>Fixed value (0)</p> <p>Fixed value (0)</p> <p>Fixed G0 inclination</p>
K97	<p>B-axis misalignment correction Name of parallel axis</p> <p>Program type M, E Conditions Immediate Unit Hexadecimal, two digits Setting range &41 to &5A</p>		<p>Specify the name of the axis to be made parallel with respect to the direction of the spindle when the angle of the B-axis is 0 degrees.</p> <p>Note : If 0 is entered, the axis will be regarded as the Z-axis (&5A).</p>
K98	<p>B-axis misalignment correction Name of orthogonal axis</p> <p>Program type M, E Conditions Immediate Unit Hexadecimal, two digits Setting range &41 to &5A</p>		<p>Specify the name of the axis to be made orthogonal with respect to the direction of the spindle when the angle of the B-axis is 0 degrees.</p> <p>Note : If 0 is entered, the axis will be regarded as the X-axis (&58).</p>

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K99	Dynamic offset Name of rotational axis		Specify the name of the rotational axis to undergo dynamic offset.
	Program type M, E Conditions Immediate Unit Hexadecimal, two digits Setting range &41 to &5A		
K100	Dynamic offset Name of parallel axis		Specify the name of the axis to be made parallel with respect to the rotational plane of the rotational axis to undergo dynamic offset.
	Program type M, E Conditions Immediate Unit Hexadecimal, two digits Setting range &41 to &5A		
K101	Dynamic offset Name of orthogonal axis		Specify the name of the axis to be made orthogonal with respect to the rotational plane of the rotational axis to undergo dynamic offset.
	Program type M, E Conditions Immediate Unit Hexadecimal, two digits Setting range &41 to &5A		
K102 (bit 0) to (bit 2)	—		Fixed value (0)
	Program type — Conditions — Unit — Setting range —		

Classification	MACHINE	Display title	MEASURE																																				
Address	Meaning	Description																																					
K102 (bit 3)	Compensation method for the sensing tool's eccentricity and stylus radius in workpiece measurement and coordinate measurement	<p>How to use the data of compensation for the sensing tool's eccentricity and stylus radius in workpiece measurement and coordinate measurement.</p> <p>0: M Pro scheme For workpiece measurement: No use of eccentricity settings; Use of stylus radius settings for both longitudinal (workpiece-axial) and transverse (workpiece-radial) directions. For coordinate measurement: Use of eccentricity setting for the direction of measuring feed; Use of stylus radius setting for transverse (workpiece-radial) direction.</p> <p>1: MATRIX scheme Full use of the eccentricity and stylus radius settings for both workpiece measurement and coordinate measurement.</p> <p>* Setting of this parameter and compensation method</p> <table border="1"> <thead> <tr> <th rowspan="2"></th> <th rowspan="2">K102 bit 3</th> <th colspan="2">Eccentricity</th> <th colspan="2">Stylus radius</th> </tr> <tr> <th>Measurement direction</th> <th>Other directions</th> <th>Radial</th> <th>Axial</th> </tr> </thead> <tbody> <tr> <td>Workpiece measurement</td> <td>0</td> <td>Invalid</td> <td>Invalid</td> <td>Valid</td> <td>Valid</td> </tr> <tr> <td></td> <td>1</td> <td>Valid</td> <td>Valid</td> <td>Valid</td> <td>Valid</td> </tr> <tr> <td>Coordinate measurement</td> <td>0</td> <td>Valid</td> <td>Invalid</td> <td>Invalid</td> <td>Valid</td> </tr> <tr> <td></td> <td>1</td> <td>Valid</td> <td>Valid</td> <td>Valid</td> <td>Valid</td> </tr> </tbody> </table>					K102 bit 3	Eccentricity		Stylus radius		Measurement direction	Other directions	Radial	Axial	Workpiece measurement	0	Invalid	Invalid	Valid	Valid		1	Valid	Valid	Valid	Valid	Coordinate measurement	0	Valid	Invalid	Invalid	Valid		1	Valid	Valid	Valid	Valid
	K102 bit 3	Eccentricity		Stylus radius																																			
		Measurement direction	Other directions	Radial	Axial																																		
Workpiece measurement	0	Invalid	Invalid	Valid	Valid																																		
	1	Valid	Valid	Valid	Valid																																		
Coordinate measurement	0	Valid	Invalid	Invalid	Valid																																		
	1	Valid	Valid	Valid	Valid																																		
Program type	M																																						
Conditions	Immediate																																						
Unit	—																																						
Setting range	0, 1																																						
Z-axis direction for setting the TPC relay points for a measurement unit	<p>Z-axis direction for setting the TPC relay points for a measurement unit.</p> <p>0: Direction opposite to the program coordinate system</p>  <p>1: Direction of the program coordinate system</p> 																																						
K102 (bit 4)	Z-axis direction for setting the TPC relay points for a measurement unit	<p>Z-axis direction for setting the TPC relay points for a measurement unit.</p> <p>0: Direction opposite to the program coordinate system</p>  <p>1: Direction of the program coordinate system</p> 																																					
	Program type	M																																					
	Conditions	Immediate																																					
	Unit	—																																					
	Setting range	0, 1																																					
	Barrier information setting for EIA/ISO programs	<p>Barrier information for EIA/ISO programs is to be set</p> <p>0: in the relevant parameters, 1: on the SET UP MANAGE. display.</p>																																					
K102 (bit 5)	Barrier information setting for EIA/ISO programs	<p>Barrier information for EIA/ISO programs is to be set</p> <p>0: in the relevant parameters, 1: on the SET UP MANAGE. display.</p>																																					
	Program type	E																																					
	Conditions	Immediate																																					
	Unit	—																																					
	Setting range	0, 1																																					

Classification	MACHINE	Display title	MEASURE
Address	Meaning	Description	
K102 (bit 6)	Setting the direction of rotation for milling tools	Whether or not, on the TOOL DATA display, the direction of rotation can be specified for milling tools mounted on the turret. 0: No 1: Yes (Normal/Reverse)	
	Program type	M	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
K102 (bit 7)	—	Fixed value (0)	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K103	—	<p>Handling of the sign assigned to hobbing address D (Note 1) 0: If plus (+), the workpiece spindle and the milling spindle rotate in opposite directions. If minus (-), both spindles rotate in the same direction. 1: If plus (+), the workpiece spindle and the milling spindle rotate in the same direction. If minus (-), both spindles rotate in opposite directions (MT Pro compatible).</p> <p>Synchronous control error auto-correction (Note 2) 0: Valid 1: Invalid</p> <p>Fixed value DDB micron unit 0: Interpolation 1: Microns</p> <p>Fixed value Output of S-code and T-code at restart 0: Invalid 1: Valid * Set 0 for machining center specifications.</p> <p>Whether FIXED PT can be selected at RETURN of the end unit of the MAZATROL program 0: Possible to select 1: Impossible to select</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	MACHINE	Display title	MEASURE								
K104	<p>—</p> <table border="1" style="margin-left: 20px;"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>Bit</td></tr> <tr><td>Setting range</td><td>Binary, eight digits</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	Bit	Setting range	Binary, eight digits	<p>Output of a laser measurement B-axis 0-degree command after ATC 0: Output invalid 1: Output valid</p> <p>Direction of the laser axis of the laser measuring instrument 0: Depends on the setting of parameter L16. L16 = 1: Parallel to the Y-axis, L16 = 2: Parallel to the X-axis 1: Depends on the setting of bit 2 in parameter K104.</p> <p>Direction of the laser axis of the laser measuring instrument 0: Parallel to the Y-axis 1: Parallel to the X-axis</p> <p>Note : This parameter is valid when K104 bit 1 = 1.</p> <p>Voice Adviser (Vocal output function) 0: Invalid 1: Valid</p> <p>Type of voice 0: Male's voice 1: Female's voice</p> <p>Vocal guidance for warm-up operation 0: Invalid 1: Valid</p> <p>Machine status when rapid feed command is given with cutting feed override set to 0% 0: Alarm stop 1: Operation is executed.</p> <p>Feed rate for an approach to the starting point in EIA modal restarting mode 0: Approach at the feed rate during manual operations 1: If the modal command during the search is G0, approach at G0, or if the modal command is other than G0, approach at G1</p>	<p>—</p> <p>—</p> <p>—</p> <p>—</p> <p>—</p> <p>—</p> <p>—</p>
Program type	M, E										
Conditions	Immediate										
Unit	Bit										
Setting range	Binary, eight digits										
K105 (bit 0)	<p>—</p> <table border="1" style="margin-left: 20px;"> <tr><td>Program type</td><td>—</td></tr> <tr><td>Conditions</td><td>—</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>—</td></tr> </table>	Program type	—	Conditions	—	Unit	—	Setting range	—	Invalid	
Program type	—										
Conditions	—										
Unit	—										
Setting range	—										

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K105 (bit 1)	 — Program type — Conditions — Unit — Setting range —		Fixed value (1)
K105 (bit 2)	S-code macro call invalid/valid Program type M, E Conditions At power on Unit — Setting range 0, 1		0: Invalid 1: Valid
K105 (bit 3)	T-code macro call invalid/valid Program type M, E Conditions At power on Unit — Setting range 0, 1		0: Invalid 1: Valid
K105 (bit 4)	Second auxiliary function macro call invalid/valid Program type M, E Conditions At power on Unit — Setting range 0, 1		0: Invalid 1: Valid

Classification	MACHINE	Display title	MEASURE
Address	Meaning	Description	
K105 (bit 5)	—	Fixed value (0)	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K105 (bit 7)	—	Fixed value (0)	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K106	—	<p>Execution conditions for user macroprogram interrupt 0: Edge triggering (Performed just once when the interrupt signal is turned ON) 1: Status triggering (Repeatedly performed while the interrupt signal is ON)</p> <p>Start timing for user macroprogram interrupt 0: The block under execution is aborted and then the interrupt occurs immediately. 1: Interrupt occurs after completion of the block being executed.</p> <p>Fixed value (0)</p>	
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K107 (bit 0)	—	Fixed value (0)	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
K107 (bit 5)	Motion distance in the manual pulse feed mode for the axis whose position is displayed in diameter value.	0: In radius values 1: In diameter values * Invalid for the axis for which bit 5 of M18 is set to 1.	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
K107 (bit 6)	Deceleration for arc valid/invalid	Deceleration for arc valid/invalid	
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	0, 1	
K107 (bit 7)	—	Fixed value (0)	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K108	Permissible error range for synchronous control		<p>Specify the maximum permissible error range for the master and slave axes for synchronous control (Tandem driving system).</p> <p>If the specified range is overstepped, alarm 141 EXCESS SIMULTANEOUS ERROR will be displayed.</p> <p>Note : Error checking will not occur if 0 is set.</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	0 to 65535	
K110	Judgment angle near a singular point (Tool tip point control)		Set the judgment angle near a singular point. The setting, if 0, will be handled as 1°.
	Program type	E	
	Conditions	After stop of axis movement	
	Unit	°	
	Setting range	0 to 360	
K111	Clamping speed in safety supervisory mode 3		Set the clamping speed (speed command) in safety supervisory mode 3.
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	mm/min	
	Setting range	0 to 1000	

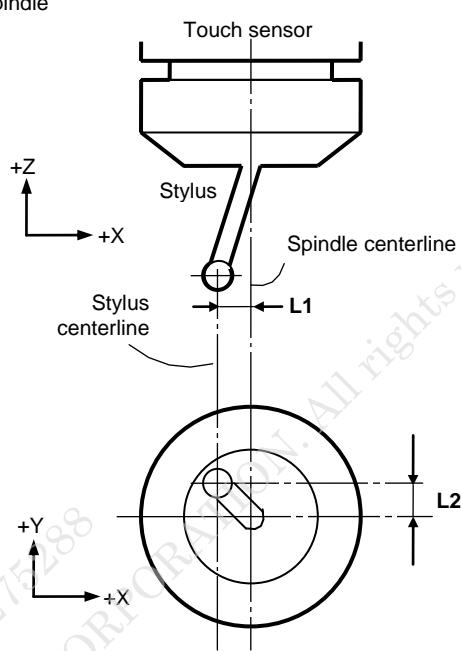
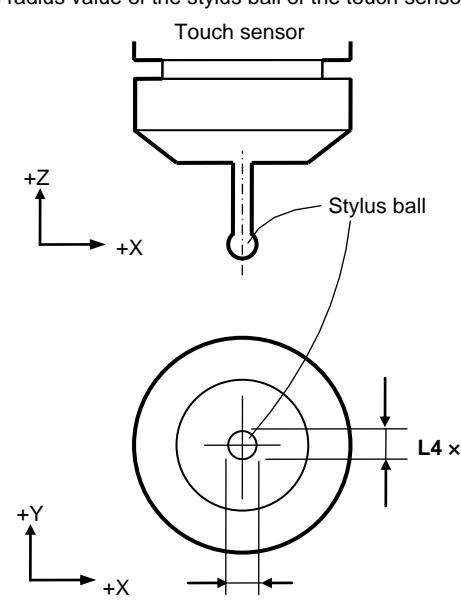
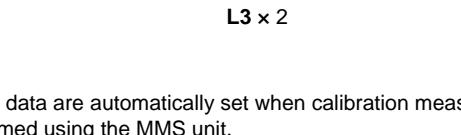
Classification	MACHINE	Display title	MEASURE							
K113	<p>Machine type (Tool tip point control)</p> <table border="1" style="margin-left: 10px;"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>1 to 3</td></tr> </table>	Program type	E	Conditions	Immediate	Unit	—	Setting range	1 to 3	<p>Set the type of machine.</p> <p>1: Tool tilt type 2: Mixed type 3: Table tilt type</p> <p>Example :</p> <p>1 = Tool tilt type</p> <p>2 = Mixed type</p> <p>3 = Table tilt type</p>
Program type	E									
Conditions	Immediate									
Unit	—									
Setting range	1 to 3									
K114	<p>Axis number of the horizontal axis in the rectangular coordinate system (Tool tip point control)</p> <table border="1" style="margin-left: 10px;"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 16</td></tr> </table>	Program type	E	Conditions	Immediate	Unit	—	Setting range	0 to 16	<p>Set the axis number of the horizontal axis in the rectangular coordinate system. A setting of 0 is invalid.</p>
Program type	E									
Conditions	Immediate									
Unit	—									
Setting range	0 to 16									
K115	<p>Axis number of the vertical axis in the rectangular coordinate system (Tool tip point control)</p> <table border="1" style="margin-left: 10px;"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 16</td></tr> </table>	Program type	E	Conditions	Immediate	Unit	—	Setting range	0 to 16	<p>Set the axis number of the vertical axis in the rectangular coordinate system. A setting of 0 is invalid.</p>
Program type	E									
Conditions	Immediate									
Unit	—									
Setting range	0 to 16									

Classification	MACHINE	Display title	MEASURE
Address	Meaning		Description
K116	Axis number of the height axis in the rectangular coordinate system (Tool tip point control)		Set the axis number of the height axis in the rectangular coordinate system. A setting of 0 is invalid.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 16	
K117	Rotational direction of the rotary axis (Tool tip point control)		12: 2nd rotary axis..... Rotates about horizontal axis, 1st rotary axis..... Rotates about vertical axis. 13: 2nd rotary axis Rotates about horizontal axis, 1st rotary axis..... Rotates about height axis. 21: 2nd rotary axis..... Rotates about vertical axis, 1st rotary axis..... Rotates about horizontal axis. 23: 2nd rotary axis..... Rotates about vertical axis, 1st rotary axis..... Rotates about height axis. 31: 2nd rotary axis..... Rotates about height axis, 1st rotary axis..... Rotates about horizontal axis. 32: 2nd rotary axis..... Rotates about height axis, 1st rotary axis Rotates about vertical axis.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	—	
K121	Axis number of the first rotary axis (Tool tip point control)		Set the axis number of the first rotary axis. A setting of 0 is invalid.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 16	
K122	Horizontal axis rotational center offset of the first rotary axis (Tool tip point control)		<If the first rotary axis rotates for tool control> Set the distance from the rotational center of the tool control rotary axis (at the tool side) in the direction of the horizontal axis to the rotational center of the tool control rotary axis (at the opposite side). <If the first rotary axis rotates for table control> Set the distance from the spindle tip point in the direction of the horizontal axis to the rotational center of the table control rotary axis (at the opposite side) when all axes are in the machine home position.
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	±99999999	

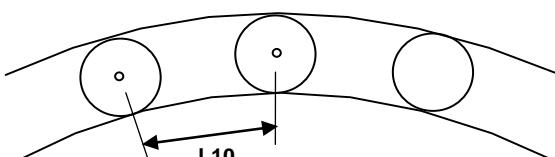
Classification	MACHINE	Display title	MEASURE
Address	Meaning	Description	
K123	Vertical axis rotational center offset of the first rotary axis (Tool tip point control)	<If the first rotary axis rotates for tool control> Set the distance from the rotational center of the tool control rotary axis (at the tool side) in the direction of the vertical axis to the rotational center of the tool control rotary axis (at the opposite side). <If the first rotary axis rotates for table control> Set the distance from the spindle tip point in the direction of the horizontal axis to the rotational center of the table control rotary axis (at the opposite side) when all axes are in the machine home position.	
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	±99999999	
K124	Height axis rotational center offset of the first rotary axis (Tool tip point control)	<If the first rotary axis rotates for tool control> Set the distance from the rotational center of the tool control rotary axis (at the tool side) in the direction of the height axis to the rotational center of the tool control rotary axis (at the opposite side). Note : In the case of machines with the capability of Mazatech, the height axis rotational center offset is equal to K124 + J121 (amount of height axis rotational center offset of the rotary axis on the tool side). <If the first rotary axis rotates for table control> Set the distance from the spindle tip point in the direction of the height axis to the rotational center of the table control rotary axis (at the opposite side) when all axes are in the machine home position. Note : When K113 = 2 (machine type: mix type), K124 has the same meaning as BA62 (amount of offset for the B-axis - spindle). In the case of inch specifications, multiply the BA62 value by 2.54, round off to the nearest whole number, and enter this value to K124.	
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	±99999999	
K125	Axis number of the second rotary axis (Tool tip point control)	Set the axis number of the second rotary axis. A setting of 0 is invalid.	
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 16	

Classification	MACHINE	Display title	MEASURE
Address	Meaning	Description	
K126	Horizontal axis rotational center offset of the second rotary axis (Tool tip point control)		<If the second rotary axis rotates for tool control> Set the distance from the spindle holder end in the direction of the horizontal axis to the rotational center of the tool control rotary axis (at the tool side). <If the second rotary axis rotates for table control> Set the distance from the rotational center of the table control rotary axis in the direction of the horizontal axis to the rotational center of the table control rotary axis (at the workpiece side) when all axes are in the machine home position.
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	±99999999	
K127	Vertical axis rotational center offset of the second rotary axis (Tool tip point control)		<If the second rotary axis rotates for tool control> Set the distance from the spindle holder end in the direction of the vertical axis to the rotational center of the tool control rotary axis (at the tool side). <If the second rotary axis rotates for table control> Set the distance from the rotational center of the table control rotary axis in the direction of the vertical axis to the rotational center of the table control rotary axis (at the workpiece side) when all axes are in the machine home position.
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	±99999999	
K128	Height axis rotational center offset of the second rotary axis (Tool tip point control)		<If the second rotary axis rotates for tool control> Set the distance from the spindle holder end in the direction of the height axis to the rotational center of the tool control rotary axis (at the tool side). <If the second rotary axis rotates for table control> Set the distance from the rotational center of the table control rotary axis in the direction of the height axis to the rotational center of the table control rotary axis (at the workpiece side) when all axes are in the machine home position.
	Program type	E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	±99999999	

2-3-11 Machine parameter TABLE (L)

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L1	Stylus eccentricity of touch sensor (X-component)	The eccentricity of the stylus of the touch sensor with respect to the center of the spindle	
	Program type	M	
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 in	
	Setting range	0 to ±99999999	
L2	Stylus eccentricity of touch sensor (Y-component)		
	Program type	M	MPL093
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 in	
	Setting range	0 to ±99999999	
Note :	<ol style="list-style-type: none"> 1. These data are automatically set when calibration measurement is performed using the MMS unit. 2. With 5-axis machines, L84 and L85 are used for these settings. 		
L3	Radius of stylus ball of touch sensor (X-component)	The true radius value of the stylus ball of the touch sensor	
	Program type	M	
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 in	
	Setting range	0 to ±99999999	
L4	Radius of stylus ball of touch sensor (Y-component)		
	Program type	M	MPL094
	Conditions	At power on	
	Unit	0.0001 mm/0.00001 in	
	Setting range	0 to ±99999999	
Note :	<p>These data are automatically set when calibration measurement is performed using the MMS unit.</p>		

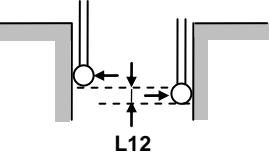
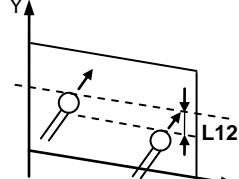
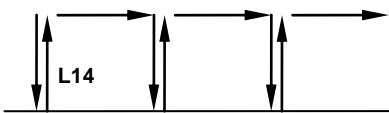
Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L5		<p>The distance from the spindle taper gage line to the table surface, No. 1 turning spindle chuck edge, or the reference block on the pallet existing when the Z-axis is in the machine zero-point position</p> <p>Z-axis stroke for tip position memory (TEACH function)</p> <p>Program type M, E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to ±99999999</p>	
L6		<p>The minimum tool displacement by which the tool is judged to be a broken one as a result of execution of the tool breakage detection function</p> <p>If (registered tool length data) – (tool length data that has been measured during the detecting operation) \geq L6, then the tool is judged broken.</p> <p>Tool-breakage judgment distance for TBR function</p> <p>Program type M, E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to ±99999999</p>	
L7		<p>The parameter for selecting the type of restoration to be performed after tool breakage has been detected as a result of execution of the external tool breakage detection function (M35)</p> <p>1: Single-block stop 2: Machining restarts from the next process. If there is only one process, the end unit is executed. 3: Single-block stop occurs in a state where machining can be restarted from the next process.</p> <p>Tool-breakage restoration mode for TBR function (for M35 external tool breakage detection)</p> <p>Program type M Conditions Immediate Unit — Setting range 1 to 3</p>	

Classification	MACHINE	Display title	TABLE								
Address	Meaning	Description									
L8	<p>Skipping stroke limit for MMS</p> <table border="1"> <tr> <td>Program type</td><td>M, E</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>0.0001 mm/0.00001 in</td></tr> <tr> <td>Setting range</td><td>0 to 9999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.0001 mm/0.00001 in	Setting range	0 to 9999999	<p>The maximum skipping movement distance for the measurement with the MMS unit An alarm message will appear if the touch sensor has not come into contact with the workpiece within this distance.</p>	
Program type	M, E										
Conditions	Immediate										
Unit	0.0001 mm/0.00001 in										
Setting range	0 to 9999999										
L9	<p>Selection of random ATC specifications</p> <table border="1"> <tr> <td>Program type</td><td>M, E</td></tr> <tr> <td>Conditions</td><td>At power on</td></tr> <tr> <td>Unit</td><td>—</td></tr> <tr> <td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M, E	Conditions	At power on	Unit	—	Setting range	0, 1	<p>Set to 1 when the machine of the random ATC specifications is used. 0: Standard machine 1: Machine of random ATC specifications</p>	
Program type	M, E										
Conditions	At power on										
Unit	—										
Setting range	0, 1										
L10	<p>Interval between magazine pockets</p> <table border="1"> <tr> <td>Program type</td><td>M, E</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>1 mm/0.1 in</td></tr> <tr> <td>Setting range</td><td>0 to 999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	1 mm/0.1 in	Setting range	0 to 999	<p>Set the interval between magazine pockets.</p>  <p>MPL511</p>	
Program type	M, E										
Conditions	Immediate										
Unit	1 mm/0.1 in										
Setting range	0 to 999										
L11	<p>Touch sensor's interference direction</p> <table border="1"> <tr> <td>Program type</td><td>M, E</td></tr> <tr> <td>Conditions</td><td>Immediate</td></tr> <tr> <td>Unit</td><td>—</td></tr> <tr> <td>Setting range</td><td>0 to 2</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0 to 2	<p>Set the touch sensor's interference direction.</p> <ul style="list-style-type: none"> 0: Non-interference (normal diameter) 1: To jut out in the direction of a pocket of higher number (Positive direction of magazine) 2: To jut out in the direction of a pocket of lower number (Negative direction of magazine) 	
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0 to 2										

Classification	MACHINE
----------------	---------

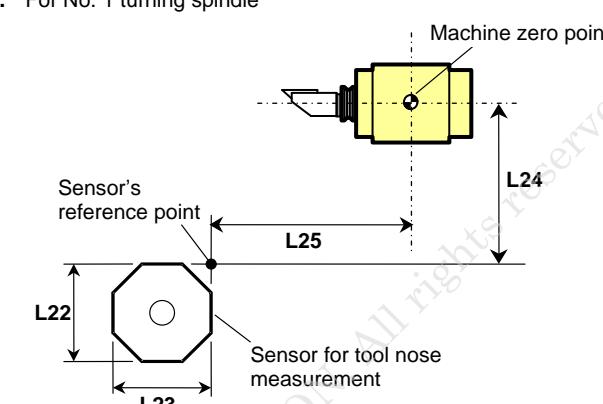
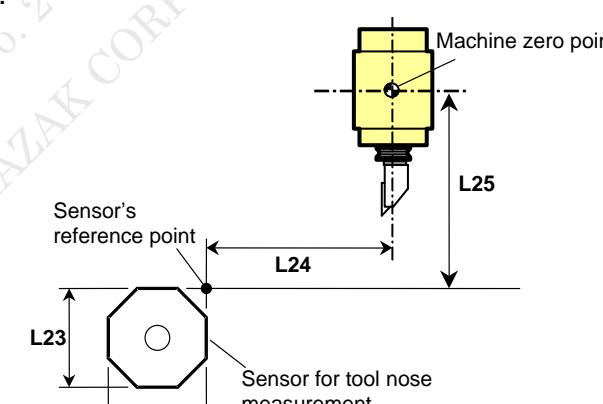
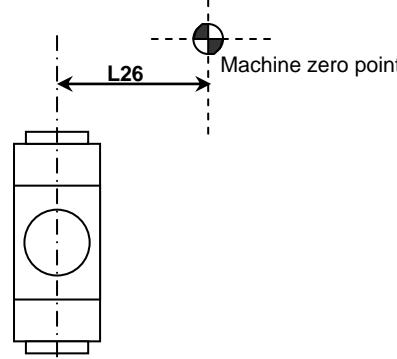
Display title

TABLE

Address	Meaning	Description
L12	Tolerance for manual measurement	Tolerance for Z coordinate value in circle measurement 
		Tolerance for measured coordinate value in face measurement 
	Program type	M, E
	Conditions	Immediate
	Unit	0.0001 mm/0.00001 in
L13	Allowable angle for parallelism and right angle in manual measurement	Set the allowable angle for calculation of parallelism and right angle to be measured manually.
	Program type	M
	Conditions	Immediate
	Unit	0.0001°
	Setting range	0 to ±900000
L14	Escapement for straightness measurement	Set an escape amount from a measurement point to the next point in straightness measurement. 
	Program type	M
	Conditions	Immediate
	Unit	0.0001 mm/0.00001 in
	Setting range	0 to ±99999999

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L15	Macro program number for straightness measurement Program type M Conditions Immediate Unit — Setting range 0 to 99999999	Set macro program number for straightness measurement. Before shipment, the macro program is numbered "9999" at the factory.	
L16	— Program type — Conditions — Unit — Setting range —	Parameter for system internal setting <u>Setting prohibited</u>	
L17	Minimum angle of head indexing command Program type M, E Conditions Next block Unit 0.1° Setting range 0 to 3600	Set a command unit of head indexing angle. (For INTEGREX j)	
L18	Presence/absence of tailstock Program type M, E Conditions Immediate Unit — Setting range 0, 1	0: Tailstock absent. 1: Tailstock present.	

Classification	MACHINE	Display title	TABLE
Address	Meaning		Description
L19	Number of electric steady rests		Set the number of electric steady rests. 0: No electric steady rest 1: One electric steady rest 2: Two electric steady rests
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	
L21	Output type of index (rotary) table		Select the output type for the angle command of the indexing unit and the end unit of the MAZATROL program. 0: To select servo axis (4th axis) 1: To select the code (the second auxiliary function) set by K56 2: To select servo axis (4th/5th axis)
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L22 to L25	<p>Data of the tool nose measurement sensor</p> <p>L22 : Sensor width along the X-axis L23 : Sensor width along the Z-axis L24 : X-coordinate of the sensor's reference point L25 : Z-coordinate of the sensor's reference point</p>	<p>Use L22 and L23 to set the dimensions of the sensor for tool nose measurement. L24 and L25 are provided to set in machine coordinates the reference point of the sensor (see the diagram below).</p> <p>Example : [INTE IV, INTE e-HII, lathes]</p> <ol style="list-style-type: none"> For No. 1 turning spindle  <p>Set the imaginary upper left corner of the sensor as its reference point for No. 2 spindle.</p> <p>[INTE e-VII]</p> <ol style="list-style-type: none"> Machine zero point 	<p>Note : As shown above, the X- and Z-axes must be replaced with each other between INTEGREX e-HII and e-VII.</p>
L26	<p>Tool nose measurement sensor reference position, Y-axis</p>	<p>Setting of sensor reference point Y coordinate</p> 	

Classification	MACHINE	Display title	TABLE
Address		Meaning	
L27		During manual TOOL EYE measurement, even if the sensor turns on for a time shorter than that which has been specified in this parameter, that will not be regarded as sensor-on.	
Timer setting for manual TOOL EYE measurement			
Program type		M, E	
Conditions		Immediate	
Unit		7.1 ms	
Setting range		0 to 65535	
L28		<p>In the machine configuration that will cause the arm of the TOOL EYE to interfere with the tool when the arm is extended or retracted with the tool present at the measuring approach point during automatic tool measurement with the TOOL EYE, set the distance through which the tool is to be returned in the Z-axis direction before extending or retracting the arm for the measurement.</p> <p>In the MAZATROL program, before the arm is extended or retracted, the tool is returned through the L28/L78-set distance from the measuring approach point.</p> <p>In the EIA/ISO program, when G136 command execution is completed, the tool will be returned through the L28/L78-set distance from the approach point. (In the EIA/ISO program, the tool will be retreated to a relay point, only in retraction mode, and the arm will be extended by execution of M284 in the next block onward.)</p> <p>(For QTN machine specifications, parameter L28 becomes valid for the #2 and #4 approach points in the -Z direction.)</p>	
Amount of Z-axisal retreat from the approach point for TOOL EYE measurement			
Program type		M, E	
Conditions		Immediate	
Unit		0.0001 mm/0.00001 in	
Setting range		±99999999	

Classification	MACHINE	Display title	TABLE										
Address	Meaning		Description										
L29	Machine efficiency Program type M Conditions Immediate Unit % Setting range 0 to 100		Set machine efficiency. This value is used as average output calculation data for machining navigation. If the setting is 0, machine efficiency will become 90%.										
L30	Selection of machining navigation case introduction messages Program type M, E Conditions Immediate Unit — Setting range 0 to 999		Select for each machine model the appropriate case introduction messages on MACHINING NAVIGATION-PREDICTION display. Specific data is preset for each machine model. Do not disturb the presettings.										
L31 to L34	Use in machine manufacturer macro Program type M, E Conditions Immediate Unit — Setting range ±99999999		The settings of these parameters can be read using system variables. <table border="1"><thead><tr><th>Parameter</th><th>System variable</th></tr></thead><tbody><tr><td>L31</td><td>#3103</td></tr><tr><td>L32</td><td>#3104</td></tr><tr><td>L33</td><td>#3105</td></tr><tr><td>L34</td><td>#3106</td></tr></tbody></table>	Parameter	System variable	L31	#3103	L32	#3104	L33	#3105	L34	#3106
Parameter	System variable												
L31	#3103												
L32	#3104												
L33	#3105												
L34	#3106												
L37	Minimum index angle of index table Program type M Conditions At power on Unit ° Setting range 0 to 180		For the command to rotate the index table, specify the minimum index angle for angle command by the M-code or B-code. Note : This parameter is ineffective for the system with an NC rotary table.										

Classification	MACHINE	Display title	TABLE								
Address	Meaning	Description									
L38	M/B-code for index of index table <table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>At power on</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 9999</td></tr> </table>	Program type	M	Conditions	At power on	Unit	—	Setting range	0 to 9999	<p>For the command to rotate the index table, select the turning direction and the M-code number to be output.</p> <p>0 or 1: B-code (0: Turning in the direction of CW, 1: Turning in the direction of CCW/shortcut) 2 to 9999: M-code (Numeric value is the M-code number to be output)</p> <p>Note :</p> <ol style="list-style-type: none"> 1. This parameter is ineffective for the system with an NC rotary table. 2. Turning direction of the index table can be selected in the indexing unit only when this parameter is set to 1. 	
Program type	M										
Conditions	At power on										
Unit	—										
Setting range	0 to 9999										
L39	Selection of execution/non execution of indexing unit <table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>At power on</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	At power on	Unit	—	Setting range	0, 1	<p>Specify the execution condition of the indexing unit. Execution of the indexing unit just before starting of machining of each tool sequence or at the end of each process:</p> <p>0: Won't be made when the index angle is the same as the preceding indexing unit execution. 1: Will be made unconditionally.</p>	
Program type	M										
Conditions	At power on										
Unit	—										
Setting range	0, 1										
L40	Availability of specification of index table angle in end unit <table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	—	Setting range	0, 1	<p>Select availability to specify the index table angle in the end unit.</p> <p>0: Enables to specify the angle. 1: Prohibits to specify the angle.</p> <p>Note : Set to 0 only for index table specification.</p>	
Program type	M										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										
L41	Simultaneous operation of indexing unit with ATC <table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>At power on</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 2</td></tr> </table>	Program type	M	Conditions	At power on	Unit	—	Setting range	0 to 2	<p>For execution of the indexing unit, specify the commanding order for movement to turning position, turning of the table and ATC.</p> <p>0: Movement to turning position → Table turning → ATC 1: Movement to turning position → Table turning and ATC 2: Movement to turning position, table turning and ATC simultaneously take place.</p> <p>Note :</p> <p>In case of setting to 2, only the X-axis coordinates can be set at turning position of the indexing unit.</p>	
Program type	M										
Conditions	At power on										
Unit	—										
Setting range	0 to 2										

Classification	MACHINE	Display title	TABLE								
Address	Meaning	Description									
L42	<p>Initial value of index table angle</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	—	Setting range	0, 1	<p>Select setting of the initial value (modal) of the index table angle for cyclic operation.</p> <p>0: Actual table angle of the machine 1: Table angle indexed at present taken as 0°</p>	
Program type	M										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										
L43	<p>Indication of index table angle</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0, 1	<p>Select showing or not showing of the index table angle on the POSITION display.</p> <p>0: Not to show 1: To show</p> <p>Note : Set to 1 for the machine with the index table, or set to 0 for that of the NC rotary table.</p>	
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										
L44	<p>Selection of automatic setting on/off for nose position correction of a drilling tool</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 2</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0 to 2	<p>Select whether automatic setting of the amount of tool nose position correction is to be made valid or invalid when entering the nose angle of a drilling tool in the tool data or when measuring the tool.</p> <p>0: Automatic setting valid When the tool nose angle is specified: The amount of correction that is appropriate for the specified angle is auto-set. When the tool nose angle is not specified: The amount of correction is not auto-set.</p> <p>1: Automatic setting invalid</p> <p>2: Automatic setting valid When the tool nose angle is specified: The amount of correction that is appropriate for the specified angle is auto-set. When the tool nose angle is not specified: The amount of correction that assumes a nose angle of 118° is auto-set.</p>	
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0 to 2										
L45	<p>Index table angle command</p> <table border="1"> <tr><td>Program type</td><td>—</td></tr> <tr><td>Conditions</td><td>—</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 8</td></tr> </table>	Program type	—	Conditions	—	Unit	—	Setting range	0 to 8	<p>Set the minimum unit of index table angle command for the indexing units, ANGLE in the end units and B-codes in the manual program unit.</p> <p>0: 1° 1 to 8: 1/1000° (MRJ2-CT specifications)</p> <p>Note : Index table angle display on the POSITION display is valid only when L43 = 1 (index table angle display on).</p> <p>0: 1° index table 1 to 7: Nth axis under MRJ2-CT specs. (N = 1 to 7) 8: Positioning table</p>	
Program type	—										
Conditions	—										
Unit	—										
Setting range	0 to 8										

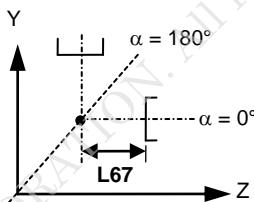
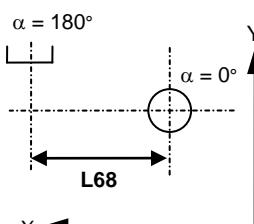
Classification	MACHINE	Display title	TABLE								
Address	Meaning	Description									
L46	<p>Maximum number of pallets in pallet changing unit</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>At power on</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 255</td></tr> </table>	Program type	M	Conditions	At power on	Unit	—	Setting range	0 to 255	<p>Select change or no change of the pallet and specify the maximum number of pallets.</p> <p>0 or 1: Not to change pallet 2 to 255: To change pallet (Numeric value indicates the maximum number of pallets.)</p> <p>Note : When this parameter is set to 0 or 1, use of the pallet changing unit is prohibited.</p>	
Program type	M										
Conditions	At power on										
Unit	—										
Setting range	0 to 255										
L47	<p>To prepare or not to prepare next pallet change</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>At power on</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	At power on	Unit	—	Setting range	0, 1	<p>Select preparation of next pallet or not to do so according to the pallet change mechanism.</p> <p>0: Not to prepare next pallet 1: To prepare next pallet</p> <p>Note : When this parameter is set to 1, it is possible to set the number of the next pallet in the pallet changing unit.</p>	
Program type	M										
Conditions	At power on										
Unit	—										
Setting range	0, 1										
L48	<p>Number of long boring bars</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 9</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0 to 9	Specify the number of long boring bars mounted.	
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0 to 9										
L49	<p>Simultaneous operation of pallet change with ATC</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	—	Setting range	0, 1	<p>This parameter is used to select simultaneous operation of pallet change with the next ATC operation in execution of the pallet changing unit and the face definition unit, or not.</p> <p>0: To operate ATC after pallet change 1: To operate pallet change and ATC simultaneously</p>	
Program type	M										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L50	Rewriting of head number Program type — Conditions Immediate Unit — Setting range 0, 1	Rewriting of head number in MDI mode: 0: Impossible 1: Possible (For five surface machining)	
L51	Tool command system in MDI operation Program type M Conditions Immediate Unit — Setting range 0, 1	Tool command system in MDI operation (Tool on the spindle and next time tool) 0: Command of pocket number 1: Command of group number	
L52	Writing of machining management data with macro variable Program type M Conditions Immediate Unit — Setting range 0, 1	Rewriting of machining management data with macro variable (system variable) 0: Impossible 1: Possible	
L53	Showing of program number in PALLET MANAGEMENT display Program type M Conditions Immediate Unit — Setting range 0, 1	Select showing or not showing of the work number in the PALLET MANAGEMENT display. 0: Not to show WNo. 1: To show WNo.	

Classification	MACHINE	Display title	TABLE
Address	Meaning		Description
L54	Selection of automatic operation mode		0: Invalid 1: FMS pallet ID operation mode 2: Pallet management operation mode
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	
L55	Spindle load meter display type		0: FR-SF 1: FR-SE
	Program type	—	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
L56	Method of measurement of coordinates by tool edge memorizing function (TEACH)		0: Method by M2 tool edge memorizing function 1: Method by M32 tool edge memorizing function (for Z-axis only) 2: Method by M32 tool edge memorizing function (for X-, Y-, Z-axes)
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	
L57	Rewriting of tool data during automatic operation		Make it possible/impossible to rewrite tool data except on tools on the spindle in automatic operation on a program. 0: Impossible 1: Possible
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L58	Head index angle indication system Program type E Conditions Immediate Unit — Setting range 0, 1	Select a head angle indication system for the five surface machining system. 0: Indication corresponding to 90° index (0°, 90°, 180°, 270°) 1: Indication corresponding to 1° (5°) index (For five surface machining)	
L59	Input selection for HEAD OFFSET display Program type M, E Conditions Immediate Unit — Setting range 0, 1	This parameter limits input items on the HEAD OFFSET display. 0: Data just on item SPDL. CMD can be input. 1: All data can be input. (For five surface machining)	
L60	Head quantity Program type M, E Conditions Immediate Unit — Setting range 0 to 10	The total number of heads to be mounted in the spindle (For five surface machining)	
L61	Output timing of AHC and APC Program type M, E Conditions Immediate Unit — Setting range 0 to 2	Operation timing of automatic head change (AHC) and automatic pallet change (APC) 0: AHC first and then APC 1: APC first and then AHC 2: Simultaneous (For five surface machining)	

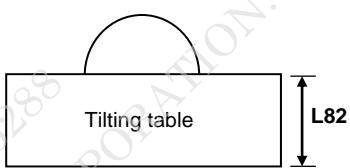
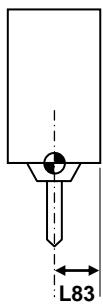
Classification	MACHINE	Display title	TABLE
Address		Meaning	Description
L62		Head relay point X1 Program type M Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to ±99999999	When FIXED is selected at the item RELAY in the face definition unit, the head arrives at the face for next machining through the point(s) specified by the parameters (to be set in the machine coordinates system). For AGX or five-surface machining models, the relay points (X1, Y1) and (X2, Y2) can be specified in the program (in the face definition sequence). For the five surface machining, the head goes through the four corners of a face where the two specified points are positioned in its diagonal line.
L63		Head relay point Y1 Program type M Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to ±99999999	<p style="text-align: right;">MPL515 (For five-surface machining) (For AGX series)</p>
L64		Head relay point X2 Program type M Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to ±99999999	
L65		Head relay point Y2 Program type M Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to ±99999999	

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L66	Return/No return to head indexing point Z Program type M Conditions Immediate Unit — Setting range 0, 1	After EIA/ISO subprogram execution; 0: Return to head indexing point Z (Even if the T-code command is for the same tool.) 1: No return to head indexing point Z (For five-surface machining) (For AGX series)	
L67	Length between the end surface of the spindle and the center of head rotation Program type M, E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to ±99999999	Set the length from the end surface of the spindle to the center of head rotation for respective machines. (100 mm in usual) 	MPL516
L68	Head correction value X Program type M, E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to ±99999999	Set for respective machines. 	MPL517 (For AGX series)
L69	Head correction value Y Program type M, E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range 0 to ±99999999	Set for respective machines. (For AGX series)	

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L70	Axis movement from machining face on escapement	<p>Specify the axes that simultaneously move from a machining face to the next machining face or in case of tool replacement.</p> <p>0: Two (three) axes simultaneously move to the safety position. 1: Y-axis (or X- and Y-axes) moves to the safety position after Z-axis moved.</p> <p>The X-axis moves when a relay point (RELAY) or a fixed point (FIXED) is selected for the item RELAY in the face definition unit.</p>	
	Program type	M	
	Conditions	Immediate	
	Unit	—	(For AGX series)
	Setting range	1, 0	
L71	Shift of basic coordinate for oblique face machining	<p>For execution of the program for oblique face machining, specify to execute or not machining on the coordinate that is turned from the basic coordinate (set in WPC unit or in OFFSET unit) at an angle of the correction value for the B-axis.</p> <p>0: Machining on the coordinate that is turned from the basic coordinate at an angle of the correction value for the B-axis 1: Machining on the basic coordinate specified in the program</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	1, 0	(For AGX series)
L73	Time constant for shape correction acceleration/deceleration filter 2	Set the time constant to be used when shape correction is on. A setting of 0 is invalid.	
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	ms	
	Setting range	0 to 56	
L74	Cutting feed rate for pre-interpolational acceleration/deceleration control	Set the cutting feed rate for pre-interpolational acceleration/deceleration control.	
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	mm/min	
	Setting range	1 to 999999	

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L75	Time constant for pre-interpolational linear control during cutting feed rate acceleration/deceleration	Set the time constant to obtain acceleration/deceleration of the cutting feed rate for pre-interpolational linear control.	
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	ms	
	Setting range	1 to 5000	
L76	Acceleration rate for high-speed cutting	Set the maximum cutting speed in the G61.1 mode at percentage to the maximum cutting speed in the G64 mode. Input of 0 is regarded as 100%. 1000 or higher percent is disposed as 1000%.	
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	%	
	Setting range	0 to 5000	
L77	Angle for deceleration at corner before interpolation	Set an angle for decelerating cutting feed rate at a corner. Input of 0 is regarded as 5°. Setting at an angle higher than 30° is disposed as 30°.	 MPL518
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	°	
	Setting range	0 to 30	

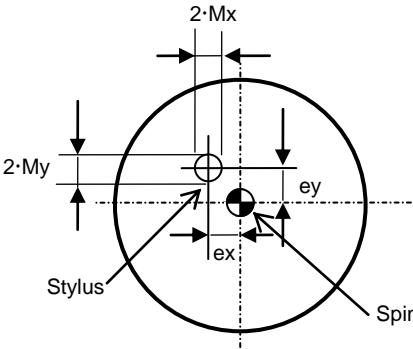
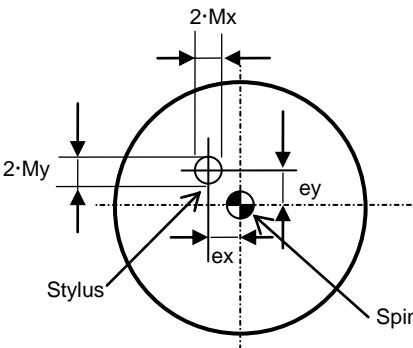
Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L78	Amount of X-axisal retreat from the approach point for TOOL EYE measurement	In the machine configuration that will cause the arm of the TOOL EYE to interfere with the tool when the arm is extended or retracted with the tool present at the measuring approach point during automatic tool measurement with the TOOL EYE, set the distance through which the tool is to be returned in the X-axis direction before extending or retracting the arm for the measurement. In the MAZATROL program, before the arm is extended or retracted, the tool is returned through the L28/L78-set distance from the measuring approach point. In the EIA/ISO program, when G136 command execution is completed, the tool will be returned through the L28/L78-set distance from the approach point. (In the EIA/ISO program, the tool will be retreated to a relay point, only in retraction mode, and the arm will be extended by execution of M284 in the next block onward.)	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 in	
	Setting range	± 99999999	
L79	In-position width for changeover of the synchronized-tapping gain	Set the In-position width for changeover of the synchronized-tapping gain. If 0 is set, 10 microns will be regarded as having been set.	
	Program type	M, E	
	Conditions	At power on	
	Unit	0.001 mm	
	Setting range	0 to 255	
L80	Linear to arc deceleration rate	Set the deceleration rate at the linear to arc or arc to linear blocks of shape correction.	
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	mm/min / in/min	
	Setting range	0 to 99999999	

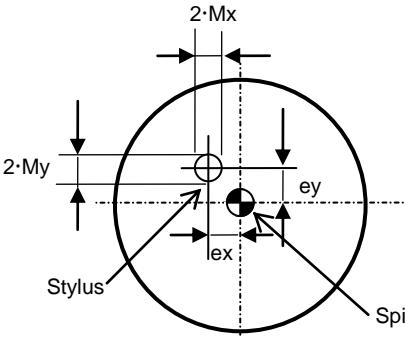
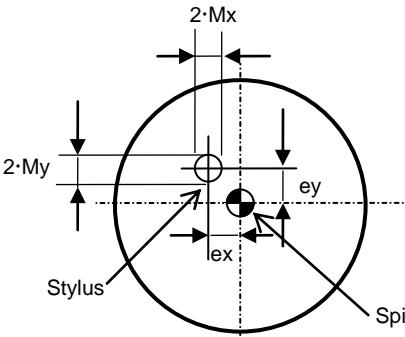
Classification	MACHINE	Display title	TABLE
Address	Meaning		Description
L81	—		<p>Parameter for system internal setting Setting prohibited</p>
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
L82	Table thickness		<p>Set the thickness of the tilting table. This parameter is used for the software travel limit function provided to avoid collision between the spindle head and the tilting table (in its angular position from -90° to -120°).</p>  <p>(For machines equipped with a tilting table)</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	± 99999999	
L83	Spindle head radius		<p>Set the radius of the spindle head. This parameter is used for the software travel limit function provided to avoid collision between the spindle head and the tilting table.</p>  <p>(For machines equipped with a tilting table)</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm	
	Setting range	± 99999999	

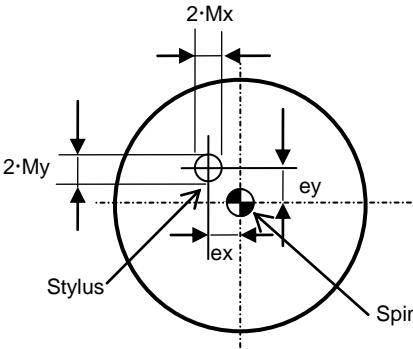
Classification MACHINE

Display title

TABLE

Address	Meaning	Description
L84	Correction value of alignment deviation X (Upper face)	 <p>2·Mx</p> <p>2·My</p> <p>Stylus</p> <p>Spindle center</p>
	Program type	M
	Conditions	After stop of movement
	Unit	0.0001 mm/0.00001 in
	Setting range	0 to ±99999999
L85	Correction value of alignment deviation Y (Upper face)	<p>ex: Alignment deviation correction value on X-axis ey: Alignment deviation correction value on Y-axis Mx: Stylus radius in the X-axis direction (The setting of L3) My: Stylus radius in the Y-axis direction (The setting of L4)</p> <p>Note : The data is set automatically by execution of calibration measurement (on the upper face) with the MMS unit.</p>
	Program type	M
	Conditions	After stop of movement
	Unit	0.0001 mm/0.00001 in
	Setting range	0 to ±99999999
L86	Correction value of alignment deviation X (0-degree face)	 <p>2·Mx</p> <p>2·My</p> <p>Stylus</p> <p>Spindle center</p>
	Program type	M
	Conditions	After stop of movement
	Unit	0.0001 mm/0.00001 in
	Setting range	0 to ±99999999
L87	Correction value of alignment deviation Y (0-degree face)	<p>ex: Alignment deviation correction value on X-axis ey: Alignment deviation correction value on Y-axis Mx: Stylus radius in the X-axis direction (The setting of L3) My: Stylus radius in the Y-axis direction (The setting of L4)</p> <p>Note : The data is set automatically by execution of calibration measurement (0-degree face) with the MMS unit.</p>
	Program type	M
	Conditions	After stop of movement
	Unit	0.0001 mm/0.00001 in
	Setting range	0 to ±99999999

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L88	<p>Correction value of alignment deviation X (90-degree face)</p> <p>Stylus eccentricity of touch sensor (X-component for B-axis at 90-degree position)</p>	 <p>ex: Alignment deviation correction value on X-axis ey: Alignment deviation correction value on Y-axis Mx: Stylus radius in the X-axis direction (The setting of L3) My: Stylus radius in the Y-axis direction (The setting of L4)</p> <p>Note : The data is set automatically by execution of calibration measurement (90-degree face) with the MMS unit.</p>	MPL519
L89	<p>Correction value of alignment deviation Y (90-degree face)</p> <p>Stylus eccentricity of touch sensor (Y-component for B-axis at 90-degree position)</p>		<p>(For INTEGREX series) (For five-surface machining)</p>
L90	<p>Correction value of alignment deviation X (180-degree face)</p> <p>Stylus eccentricity of touch sensor (X-component for B-axis at 180-degree position)</p>	 <p>ex: Alignment deviation correction value on X-axis ey: Alignment deviation correction value on Y-axis Mx: Stylus radius in the X-axis direction (The setting of L3) My: Stylus radius in the Y-axis direction (The setting of L4)</p> <p>Note : The data is set automatically by execution of calibration measurement (180-degree face) with the MMS unit.</p>	MPL519
L91	<p>Correction value of alignment deviation Y (180-degree face)</p> <p>Stylus eccentricity of touch sensor (Y-component for B-axis at 180-degree position)</p>		<p>(For INTEGREX series) (For five-surface machining)</p>

Classification	MACHINE	Display title	TABLE								
Address	Meaning	Description									
L92	<p>Correction value of alignment deviation X (270-degree face)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001 in</td></tr> <tr><td>Setting range</td><td>0 to ±99999999</td></tr> </table>	Program type	M	Conditions	After stop of movement	Unit	0.0001 mm/0.00001 in	Setting range	0 to ±99999999	 <p>ex: Alignment deviation correction value on X-axis ey: Alignment deviation correction value on Y-axis Mx: Stylus radius in the X-axis direction (The setting of L3) My: Stylus radius in the Y-axis direction (The setting of L4)</p> <p>Note : The data is set automatically by execution of calibration measurement (270-degree face) with the MMS unit.</p>	MPL519
Program type	M										
Conditions	After stop of movement										
Unit	0.0001 mm/0.00001 in										
Setting range	0 to ±99999999										
L93	<p>Correction value of alignment deviation Y (270-degree face)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001 in</td></tr> <tr><td>Setting range</td><td>0 to ±99999999</td></tr> </table>	Program type	M	Conditions	After stop of movement	Unit	0.0001 mm/0.00001 in	Setting range	0 to ±99999999		(For five-surface machining)
Program type	M										
Conditions	After stop of movement										
Unit	0.0001 mm/0.00001 in										
Setting range	0 to ±99999999										
L94	<p>X/Y travel distance during EIA-programmed tool measurement</p> <table border="1"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>	Program type	E	Conditions	Immediate	Unit	—	Setting range	0, 1	<p>The X/Y travel distance can be set during the selection of EIA-programmed fully automatic tool length measurement with the measuring table.</p> <p>0: Invalid 1: Valid</p> <p>Note : It is invalid to set this parameter under laser tool measuring equipment specifications.</p>	
Program type	E										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										
L95	<p>Offset number auto-setting for EIA-programmed tool measurement</p> <table border="1"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0, 1</td></tr> </table>	Program type	E	Conditions	Immediate	Unit	—	Setting range	0, 1	<p>The offset number is auto-set during EIA-programmed tool measurement data setting.</p> <p>0: Invalid 1: Valid</p> <p>Note : When making the auto-setting function valid, see the description of L96.</p>	
Program type	E										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										

Classification	MACHINE	Display title	TABLE								
Address	Meaning	Description									
L96	<p>Amount of shifting during offset number auto-setting for EIA-programmed tool measurement</p> <table border="1"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 4000</td></tr> </table>	Program type	E	Conditions	Immediate	Unit	—	Setting range	0 to 4000	<p>The amount of shifting for TNo. during offset number auto-setting for EIA-programmed tool measurement data setting. $[Offset\ No.] = [TNo.\ setting] + [L96\ setting]$</p> <p>Note : Valid only when L95 = 1.</p>	
Program type	E										
Conditions	Immediate										
Unit	—										
Setting range	0 to 4000										
L97	<p>M-code to be output when selecting menu function for the MDI tool designation</p> <table border="1"> <tr><td>Program type</td><td>—</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	—	Conditions	Immediate	Unit	—	Setting range	0 to 99999999	<p>1 - 999: M-code of the specified number 1000: None Other than above: M149</p>	
Program type	—										
Conditions	Immediate										
Unit	—										
Setting range	0 to 99999999										
L98	<p>Max. tool length for laser tool length measurement</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>At power on</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001 in</td></tr> <tr><td>Setting range</td><td>99999999</td></tr> </table>	Program type	M, E	Conditions	At power on	Unit	0.0001 mm/0.00001 in	Setting range	99999999	<p>Specify the maximum tool length for the MDI laser tool length measurement.</p>	
Program type	M, E										
Conditions	At power on										
Unit	0.0001 mm/0.00001 in										
Setting range	99999999										
L99	<p>Cycle time for saving the operational status management data</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>min</td></tr> <tr><td>Setting range</td><td>-1 to 1439</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	min	Setting range	-1 to 1439	<p>Specify the cycle time at which the operational status management data for the day is to be saved as a file on the hard disk.</p> <p>Note :</p> <ol style="list-style-type: none"> 1. If the setting is 0, the data will be saved each minute. 2. If the setting is -1, the data will be saved only when the date changes or when NC power is turned off. 	
Program type	M, E										
Conditions	Immediate										
Unit	min										
Setting range	-1 to 1439										

Classification	MACHINE	Display title	TABLE								
L100 to L105	<p>Meaning</p> <p>L100 Laser sensor position X L101 Laser sensor position Y L102 Laser sensor position Z L103 Approach point X for laser tool diameter measurement L104 Approach point Y for laser tool diameter measurement L105 Approach point Z for laser tool length measurement</p>	<p>[INTE e-HII]</p> <p>Approach point for tool diameter measurement</p> <p>L103: X</p> <p>L100: X</p> <p>Machine zero point</p> <p>L102: Z</p> <p>L101: Y</p> <p>L98</p> <p>L105: Z</p> <p>Approach point for tool length measurement</p> <p>[INTE e-VII]</p> <p>Machine zero point</p> <p>L100: X</p> <p>L104: Y</p> <p>L101: Y</p> <p>L98</p> <p>L105: Z</p> <p>L102: Z</p> <p>Approach point for tool diameter measurement</p> <p>Approach point for tool length measurement</p>	<p>Description</p>								
	<table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>At power on</td> </tr> <tr> <td>Unit</td> <td>0.0001 mm/0.00001 in</td> </tr> <tr> <td>Setting range</td> <td>± 99999999</td> </tr> </table>	Program type	M, E	Conditions	At power on	Unit	0.0001 mm/0.00001 in	Setting range	± 99999999		
Program type	M, E										
Conditions	At power on										
Unit	0.0001 mm/0.00001 in										
Setting range	± 99999999										
L106 (bit 0)	<p>Measuring equipment selection</p> <p>Program type M, E Conditions Immediate Unit — Setting range 0, 1</p>		<p>Select the measuring equipment to be used to perform MDI tool measurements and EIA tool measurements.</p> <p>0: Measuring table 1: Laser tool measuring equipment</p>								

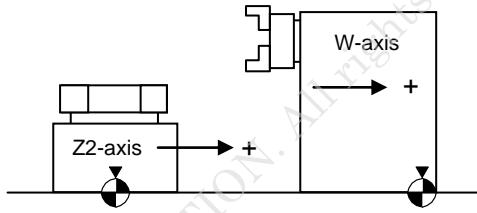
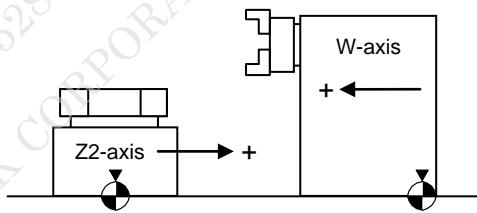
Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L106 (bit 1)	Selection of a rotational reference coordinate system for WPC-th	Select a rotational reference coordinate system for WPC-th. 0: Workpiece coordinates (Index angle B) 1: Machine coordinates	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L106 (bit 2)	Selection of measuring equipment	0: Measuring table (see L22 to L26) 1: TOOL EYE (see BA95 to BA102)	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L106 (bit 3)	—	Select whether workpiece measurement results and tool measurement results are to be stored into tool data of the lower turret. 0: Measurement results are stored into lower-turret tool data 1: Measurement results are not stored into lower-turret tool data	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L106 (bit 4)	—	Specify whether to enable or disable the fixed amount compensation function. 0: Disabled 1: Enabled	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	MACHINE	Display title	TABLE	
Address	Meaning	Description		
L106 (bit 5)	B-axis user-defined angle setting in the coordinate measurement unit and workpiece measurement unit		Select whether the user-defined angle setting for the B-axis in the coordinate measurement unit and workpiece measurement unit is to be made valid or invalid. 0: Invalid 1: Valid	
	Program type	M		
	Conditions	Immediate		
	Unit	—		
	Setting range	0, 1		
L106 (bit 6)	Non-compensation area for the tool measurement and execution timing of tool and workpiece measurement operation		Set the non-compensation area for the tool measurement and execution timing of tool and workpiece measurement operation. 0: MT Pro scheme <u>Non-compensation area setting</u> Non-compensation area for the tool measurement based on the measuring tolerance (K17 and K18) invalid (handled as if K17 = 0 and K18 = 100) <u>Execution timing of tool and workpiece measurement operation</u> The measurement unit is executed whenever the parts count reaches same or a multiple of the specified measurement interval in addition to the first time. 1: M Pro scheme <u>Non-compensation area setting</u> Non-compensation area for the tool measurement based on the measuring tolerance (K17 and K18) valid <u>Execution timing of tool and workpiece measurement operation</u> The measurement unit is executed whenever the program execution count reaches same or a multiple of the specified measurement interval, omitting the first time.	
	Program type	M, E		
	Conditions	Immediate		
	Unit	—		
	Setting range	0, 1		
L106 (bit 7)	Selection of laser measuring menu display		Select whether the laser measuring menu is to be displayed when laser tool measuring equipment is selected (L106 bit 0 = 1) as tool measuring equipment. 0: Display of the laser measuring menu 1: No display of the laser measuring menu	
	Program type	M, E		
	Conditions	Immediate		
	Unit	—		
	Setting range	0, 1		

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L107 (bit 0)	Tool path drawing	<p>In AGX machines, the tool locus (on the TRACE or TOOL PATH CHECK display) on the following coordinate is drawn by the EIA/ISO program.</p> <p>0: Loft on the standard coordinates system 1: Loft on the machine coordinates system</p> <p>Note : When 1 (drawing on the machine coordinate) is selected, the loft does not correspond to the form made on the MAZATROL coordinates system. However, 1 makes a loft corresponding to the tool movement of the machine.</p> <p>(For AGX series)</p>	
L107 (bit 1)	Tailstock function valid/invalid	Set whether the tailstock function is to be made valid or invalid. 0: Tailstock function invalid 1: Tailstock function valid	
L107 (bit 2)	Tailstock type	Select whether the tailstock included in the tailstock-equipped machine specifications is of the conventional type or the motor-driven type. 0: Conventional type 1: Motor-driven type	
L107 (bit 3)	Whether tail thrust is also to be displayed in pounds (lbs)	Select whether the tail thrust is also to be displayed in pounds (lbs). 0: Display off 1: Display on	

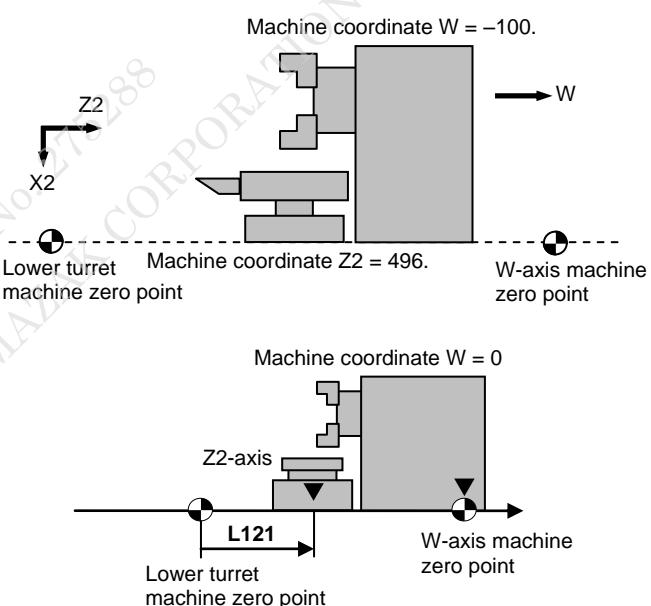
Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L107 (bit 4)	—	LBB No. setting for the grooving I.D. tool, threading I.D. tool, or touch sensor 0: Invalid 1: Valid	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L107 (bit 5)	XYZ-axis operation for the first T-command after cycle start	0: Moves the X-axis to the third zero point and the Y/Z-axes to the respective zero points. 1: Moves the X-axis to the third zero point and the Y/Z-axes to the respective second zero points.	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	(For AGX series)
L107 (bit 6)	A-axis operation for the first T-command after cycle start	0: Does not operate the A-axis. 1: Returns the A-axis to its zero point.	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	(For AGX series)
L107 (bit 7)	Heavy tool designation valid/invalid	0: Heavy tool designation valid 1: Heavy tool designation invalid	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

Classification	MACHINE	Display title	TABLE
Address	Meaning		Description
L108 (bit 0) to (bit 4)	 — Program type — Conditions — Unit — Setting range —		Fixed value (0)
L108 (bit 5)	 Incorporating the C-axis coordinate into the TOOL PATH CHECK , TRACE , and SHAPE CHECK displays of the VRX machine Program type M, E Conditions Immediate Unit — Setting range 0, 1		Set whether the C-axis coordinate obtained by C-axis coordinate rotation in the indexing unit is to be incorporated into the path and shape drawings on the TOOL PATH CHECK , TRACE , and SHAPE CHECK displays of the VRX machine. 0: Incorporation valid. 1: Incorporation invalid (M640M-5X compatible).
L108 (bit 6)	 — Program type — Conditions — Unit — Setting range —		Fixed value (0)
L108 (bit 7)	 Displaying the milling tool nose in the MAZATROL program of the TOOL PATH CHECK display Program type M Conditions Immediate Unit — Setting range 0, 1		Set whether, during tool path check of the MAZATROL program on the TOOL PATH CHECK display of a machine having a turning function, the tool nose of the milling tool is to be displayed with the tool diameter similarly to the M640M Pro or with a tool nose mark similarly to the M640MT Pro. 0: Display with tool diameter (M640M Pro compatible) 1: Display with tool nose mark (M640MT Pro compatible) Note : 1. A tool nose mark is always displayed when the tool path of the EIA/ISO program is checked. 2. For a machine without a turning function, this parameter is invalid and the display is always made with the tool diameter.

Classification	MACHINE	Display title	TABLE								
Address	Meaning	Description									
L109 (bit 0) to (bit 3)	<p>Axial direction for checking the software limit relative position Bit 0: Axial direction (1st set of axes) Bit 1: Axial direction (2nd set of axes) Bit 2: Axial direction (3rd set of axes) Bit 3: Axial direction (4th set of axes)</p>	<p>Specify the relationship between axial directions of the two axes, on the machine coordinate system, for which software limit relative position is checked.</p> <p>0: Same direction 1: Reverse directions</p> <p>Example :</p> <p>When checking the software limit relative position between the Z2- and W-axis, the positive (+) direction of the W-axis on the machine coordinate system may be the same or reverse direction of the positive (+) direction of the Z2-axis depending on the machine model.</p> <p>Set "0" if the positive (+) direction of the Z2- and W-axis is the same on the machine coordinate system as shown in Fig. 1. Conversely, if the direction is reverse as shown in Fig. 2, then set "1".</p> <p>Fig. 1</p>  <p>Fig. 2</p> 									
	<table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>After stop of movement</td> </tr> <tr> <td>Unit</td> <td>—</td> </tr> <tr> <td>Setting range</td> <td>0, 1</td> </tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	—	Setting range	0, 1		
Program type	M, E										
Conditions	After stop of movement										
Unit	—										
Setting range	0, 1										
L109 (bit 4)	<p>Formula selection for calculating horsepower and thrust on the TOOL DATA display with the [AUTO SET] menu function</p>	<p>Selection of formula to be used for calculating the horsepower and thrust on the TOOL DATA display with the [AUTO SET] menu function</p> <p>0: M640M compatible scheme 1: MATRIX scheme</p>									
	<table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>—</td> </tr> <tr> <td>Setting range</td> <td>0, 1</td> </tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0, 1		
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0, 1										

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L110	—		
	Program type	M, E	
	Conditions	Immediate (Bit 2: At power on)	
	Unit	Bit	
	Setting range	Binary, eight digits	
L111 (bit 5)	T-code in the index unit to be executed before the pallet change unit	0: Output 1: Do not output	
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
L112	—		
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	MACHINE	Display title	TABLE								
L113 to L116	<p>Software limit relative position check – Check reference axis</p> <p>L113 1st set of axes L114 2nd set of axes L115 3rd set of axes L116 4th set of axes</p> <table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>—</td> </tr> <tr> <td>Setting range</td> <td>0 to 16</td> </tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0 to 16	<p>Set the axis to be taken as the check reference in software limit check in the approaching or separating direction. Specify the axis number by that of NC.</p> <p>The checking function will be invalid if the number is missing or if an invalid number is set.</p>	
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0 to 16										
L117 to L120	<p>Software limit relative position check – Check target axis</p> <p>L117 1st set of axes L118 2nd set of axes L119 3rd set of axes L120 4th set of axes</p> <table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>—</td> </tr> <tr> <td>Setting range</td> <td>0 to 16</td> </tr> </table>	Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0 to 16	<p>Set the check target axis corresponding to the axis set as the check reference (parameter L113 to L116) for checking the software limit relative position. Specify the axis number by that of NC.</p> <p>The checking function will be invalid if the number is missing or if an invalid number is set.</p>	
Program type	M, E										
Conditions	Immediate										
Unit	—										
Setting range	0 to 16										

Classification	MACHINE	Display title	TABLE		
Address	Meaning	Description			
L121 to L124	<p>Software limit relative position check – Limit distance in the approaching direction</p> <p>L121 1st set of axes L122 2nd set of axes L123 3rd set of axes L124 4th set of axes</p>				
	<p>For arbitrary two axes that move on the same slideway, the limit of distance those axes can approach with each other is set to prevent interference. If they are going to move closer beyond the distance set at this parameter, software limit alarm is output and they stop moving. Using parameters L121 to L124, the limit distance in the approaching direction can be set for up to four axes.</p> <p>Set the machine coordinate of the check target axis, set at parameters L117 to L120, existing at the permissible closest position from the check reference axis, set at parameters L113 to L116, which is located at the home position.</p> <p>Example:</p> <p>In the example shown below, if the W-axis and the Z2-axis are defined as the check reference axis and the check target axis, respectively, move the Z2-axis close to the W-axis, which is located at the home position, up to the position just before they interfere with each other, and set the machine coordinate of the Z2-axis located at that position. If zero is entered, the checking function in the approaching direction will be invalid. Note that the check is possible only for the combination of NC axes.</p> 				
	Program type	M, E			
	Conditions	Immediate			
	Unit	0.001 mm/0.0001 in			
L125	Setting range				
	± 99999999				
	Rotational axis prefILTERING time constant				
	Specify the time constant for smoothing the tool attitude during tool tip point control.				
	This time constant is valid only when all the following conditions are satisfied:				
	<ul style="list-style-type: none"> • High-speed smooth interpolation is valid (G05P02, G61.1, F3 bit 0 = 1) • Rotational axis prefILTERING is valid (F36 bit 7 = 1) • Tool tip point control (G43.4, G43.5) is on • Cutting feed is on 				
	Program type	E			
	Conditions	Immediate			
	Unit	ms			
	Setting range	0 to 200			
Even when the above conditions are met, if the setting of L125 is 0, rotational axis prefILTERING will be invalid.					

Classification	MACHINE	Display title	TABLE																
Address	Meaning	Description																	
L126		<p>Positioning direction of the α-axis for oblique plane indexing specified in ANGLE (plane angle) of the surface definition sequence.</p> <p>0: Makes the sign of the plane angle value invalid and positions the α-axis in its minus direction.</p> <p>1: Makes the sign of the plane angle value valid and if the plane angle is plus, positions the α-axis in its plus direction or if the plane angle is minus, positions the α-axis in its minus direction.</p> <p>2: Makes the sign of the plane angle value invalid and positions the α-axis in its plus direction.</p> <p>If the setting of this parameter is other than the above, the value of L126 will be handled as 1.</p> <p><Example of operation with 0 assigned to L126 and 45° as a plane angle> Head rotation α-axis = -114.4698° Table rotation B-axis = 65.5302°</p> <p>(For AGX series)</p>																	
<table border="1"> <tr> <td>Program type</td> <td>M, E</td> </tr> <tr> <td>Conditions</td> <td>Immediate</td> </tr> <tr> <td>Unit</td> <td>—</td> </tr> <tr> <td>Setting range</td> <td>0 to 2</td> </tr> </table>		Program type	M, E	Conditions	Immediate	Unit	—	Setting range	0 to 2										
Program type	M, E																		
Conditions	Immediate																		
Unit	—																		
Setting range	0 to 2																		
L129 L130		<p>Acceleration/deceleration filter (1st stage)</p> <p>L129 G1 time constant L130 G0 time constant</p> <p>This parameter functions as a filter to smoothen the waveform command specified for pre-interpolation acceleration/deceleration.</p> <p><Type of feed and acceleration/deceleration></p> <p>L = Post-interpolation Linear acceleration/deceleration S = Pre-interpolation S-shaped acceleration/deceleration</p> <table border="1"> <tr> <td></td> <td>G64 mode</td> <td>G61.1 mode</td> </tr> <tr> <td rowspan="3">Auto-matic</td> <td>G0</td> <td>L</td> <td>S</td> </tr> <tr> <td>G1</td> <td>L</td> <td>S</td> </tr> <tr> <td>Positioning to fixed points (for ATC, APC, etc.)</td> <td>L</td> <td>L</td> </tr> <tr> <td>Manual</td> <td>Axis feed</td> <td>L</td> </tr> </table> <p style="text-align: center;">Linear acc. & dec. S-shaped acc. & dec.</p>			G64 mode	G61.1 mode	Auto-matic	G0	L	S	G1	L	S	Positioning to fixed points (for ATC, APC, etc.)	L	L	Manual	Axis feed	L
	G64 mode	G61.1 mode																	
Auto-matic	G0	L	S																
	G1	L	S																
	Positioning to fixed points (for ATC, APC, etc.)	L	L																
Manual	Axis feed	L																	
L131 L132		<p>Acceleration/deceleration filter (2nd stage)</p> <p>L131 G1 time constant L132 G0 time constant</p> <p>Program type M, E</p> <p>Conditions After stop of axis movement</p> <p>Unit ms</p> <p>Setting range 0 to 200</p>																	

Classification	MACHINE	Display title	TABLE							
Address	Meaning	Description								
L133	<p><For tool tilt type> Tool rotational axis offset horizontal axis offset amount for the 1st rotational axis</p> <table border="1"> <tr> <td>Program type</td><td>—</td></tr> <tr> <td>Conditions</td><td>After stop of axis movement</td></tr> <tr> <td>Unit</td><td>0.0001 mm</td></tr> <tr> <td>Setting range</td><td>-30000 to 30000</td></tr> </table>	Program type	—	Conditions	After stop of axis movement	Unit	0.0001 mm	Setting range	-30000 to 30000	<p><For tool tilt type (Example : VERSATECH)> Set the distance along the horizontal axis from the rotational center of the tool-side rotational axis (tool side) to the rotational center of the tool-side rotational axis (opposite side). If the upper- or lower-limit value is overstepped, operation will be the same as that which occurs when the upper- or lower-limit value is set.</p>
Program type	—									
Conditions	After stop of axis movement									
Unit	0.0001 mm									
Setting range	-30000 to 30000									
L134	<p><For tool tilt type> Tool rotational axis offset vertical axis offset amount for the 1st rotational axis</p> <p><For table tilt type> Y-axis runout of the A-axis</p> <table border="1"> <tr> <td>Program type</td><td>—</td></tr> <tr> <td>Conditions</td><td>After stop of axis movement</td></tr> <tr> <td>Unit</td><td>0.0001 mm</td></tr> <tr> <td>Setting range</td><td>See the right column.</td></tr> </table>	Program type	—	Conditions	After stop of axis movement	Unit	0.0001 mm	Setting range	See the right column.	<p><For tool tilt type (Example : VERSATECH)> Set the distance along the vertical axis from the rotational center of the tool-side rotational axis (tool side) to the rotational center of the tool-side rotational axis (opposite side). Setting range: -30000 to 30000 If the upper- or lower-limit value is overstepped, operation will be the same as that which occurs when the upper- or lower-limit value is set.</p> <p><For table tilt type (Example : VARIAxis)> Enter the offset between the actual central position of rotation of the A-axis and the reference position in the Y-axis direction. Enter the offset with a minus sign if actual central position Y of rotation of the A-axis is present at the plus side with respect to reference position Y. Setting range: -10000 to 10000 If the upper- or lower-limit value is overstepped, operation will be the same as that which occurs when the upper- or lower-limit value is set.</p>
Program type	—									
Conditions	After stop of axis movement									
Unit	0.0001 mm									
Setting range	See the right column.									
L135	<p><For tool tilt type> Tool rotational axis offset height axis offset amount for the 1st rotational axis</p> <p><For tool tilt type> Z-axis runout of the A-axis</p> <table border="1"> <tr> <td>Program type</td><td>—</td></tr> <tr> <td>Conditions</td><td>After stop of axis movement</td></tr> <tr> <td>Unit</td><td>0.0001 mm</td></tr> <tr> <td>Setting range</td><td>See the right column.</td></tr> </table>	Program type	—	Conditions	After stop of axis movement	Unit	0.0001 mm	Setting range	See the right column.	<p><For tool tilt type (Example : VERSATECH)> Set the distance along the height axis from the rotational center of the tool-side rotational axis (tool side) to the rotational center of the tool-side rotational axis (opposite side). Setting range: -30000 to 30000 If the upper- or lower-limit value is overstepped, operation will be the same as that which occurs when the upper- or lower-limit value is set.</p> <p><For table tilt type (Example : VARIAxis)> Enter the offset between the actual central position of rotation of the A-axis and the reference position in the Z-axis direction. Enter the offset with a minus sign if actual central position Z of rotation of the A-axis is present at the plus side with respect to reference position Z. Setting range: -10000 to 10000 If the upper- or lower-limit value is overstepped, operation will be the same as that which occurs when the upper- or lower-limit value is set.</p>
Program type	—									
Conditions	After stop of axis movement									
Unit	0.0001 mm									
Setting range	See the right column.									

Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	
L136	<For tool tilt type> Tool rotational axis offset horizontal axis offset amount for the 2nd rotational axis		<For tool tilt type only (Example : VERSATECH)> Set the distance along the horizontal axis from the spindle edge to the rotational center of the tool-side rotational axis (tool side).
	Program type	—	
	Conditions	After stop of axis movement	
	Unit	0.0001 mm	
	Setting range	-30000 to 30000	
L137	<For tool tilt type> Tool rotational axis offset vertical axis offset amount for the 2nd rotational axis		<For tool tilt type only (Example : VERSATECH)> Set the distance along the vertical axis from the spindle edge to the rotational center of the tool-side rotational axis (tool side).
	Program type	—	
	Conditions	After stop of axis movement	
	Unit	0.0001 mm	
	Setting range	-30000 to 30000	
L138	<For tool tilt type> Tool rotational axis offset height axis offset amount for the 2nd rotational axis		<For tool tilt type only (Example : VERSATECH)> Set the distance along the height axis from the spindle edge to the rotational center of the tool-side rotational axis (tool side).
	Program type	—	
	Conditions	After stop of axis movement	
	Unit	0.0001 mm	
	Setting range	-30000 to 30000	
L139	Minimum tool diameter of the measurable chamfering tool (For fully automatic tool measurement with measuring table)		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.1 mm/0.01 in	
	Setting range	0 to 999	

Classification	MACHINE	Display title	TABLE
Address	Meaning		Description
L140	Waiting time for the completion signal of M35 (tool breakage detection) from PLC	0: 50 ms 1: 50 ms 2: 100 ms 3: 150 ms ⋮ ⋮ ⋮	
	Program type	M, E	
	Conditions	Immediate	
	Unit	50 ms	
	Setting range	0 to 99999999	
L141	Software limit relative position check (for MR-J2-CT axis) – Task delay period		
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	0.1 ms	
	Setting range	0 to 99999999	
L142	Software limit relative position check (for MR-J2-CT axis) – MR-J2-CT gain		
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	1/s	
	Setting range	0 to 99999999	
L143	Software limit relative position check (for MR-J2-CT axis) – MR-J2-CT time constant		
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	ms	
	Setting range	0 to 99999999	

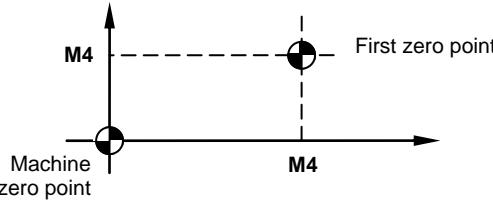
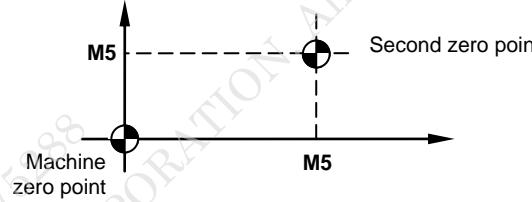
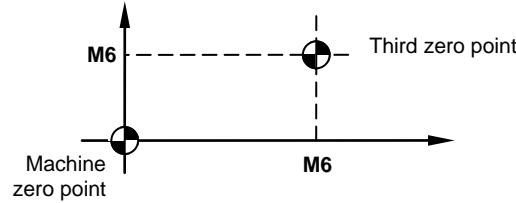
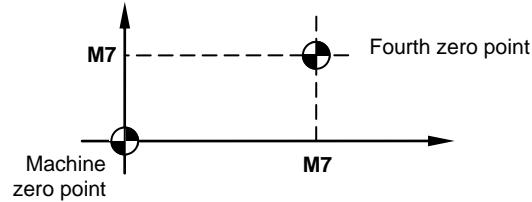
Classification	MACHINE	Display title	TABLE
Address	Meaning	Description	

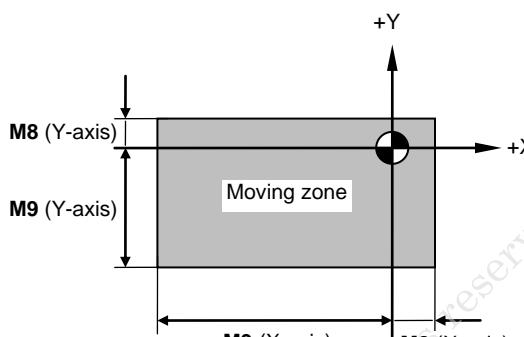
L144 Software limit relative position check (for MR-J2-CT axis) – MR-J2-CT rapid feed rate

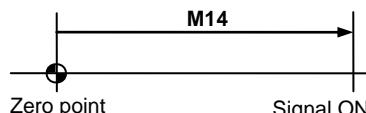
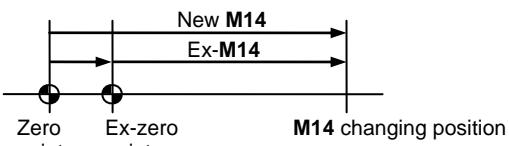
Program type	M, E
Conditions	After stop of axis movement
Unit	mm/min
Setting range	0 to 99999999

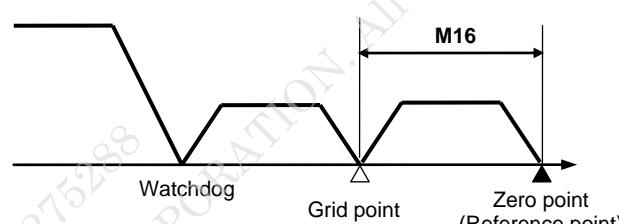
2-3-12 Machine parameter FEED VEL. (M)

Classification	MACHINE	Display title	FEED VEL.
Address	Meaning		Description
M1	Rapid feed rate		<p>The feed rate for moving each axis under the G00 command during automatic operation The feed rate for moving each axis in either the manual rapid feed mode or the zero-point return mode</p> <p>Note : Initial zero-point return is performed at the feed rate set in parameter M2.</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	mm/min (°/min)	
	Setting range	0 to 600000	
M2	Feed rate for initial zero-point return		<p>The feed rate for moving each axis during initial zero-point return (reference-point return) at power on</p> <p>Feed rate</p> <p>M2</p> <p>N8</p> <p>Watchdog</p> <p>Zero point (reference point)</p> <p>MPL520</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	mm/min (°/min)	
	Setting range	0 to 600000	
M3	Cutting feed rate limit		<p>The limit of cutting feed rate during automatic operation Even if a feed rate higher than this parameter setting is specified, the latter governs.</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	mm/min (°/min)	
	Setting range	0 to 600000	

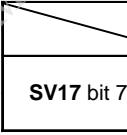
Classification	MACHINE	Display title	FEED VEL.
Address	Meaning	Description	
M4	Offset of machine coordinates system Program type M, E Conditions At power on Unit 0.0001 mm Setting range ± 99999999	The machine coordinating values of the point to which each axis is to move back under G28 command (first zero-point return).  MPL521	
M5	Second zero-point coordinating value Program type M, E Conditions After stop of movement Unit 0.0001 mm Setting range ± 99999999	The machine coordinating values of the point to which each axis is to move back under the G30 command (second zero-point return).  MPL521	
M6	Third zero-point coordinating value Program type M, E Conditions After stop of movement Unit 0.0001 mm Setting range ± 99999999	The machine coordinating values of the point to which each axis is to move back under the G30P3 command (third zero-point return).  MPL521	
M7	Fourth zero-point coordinating value Program type M, E Conditions After stop of movement Unit 0.0001 mm Setting range ± 99999999	The machine coordinating values of the point to which each axis is to move back under the G30P4 command (fourth zero-point return).  MPL521	

Classification	MACHINE	Display title	FEED VEL.																								
M8	Maximum software limit specified by manufacturer (+ direction)	The maximum moving zone permissible under the machine specifications Set the machine coordinate values. Example :																									
	Program type M, E																										
	Conditions After stop of movement																										
	Unit 0.0001 mm/0.0001°																										
	Setting range ±99999999																										
M9	Maximum software limit specified by manufacturer (– direction)		Note : This parameter is invalid when M8 = M9 . Copyright © 2013 YAMAZAKI CORPORATION. All rights reserved. Serial No. 275288																								
	Program type M, E																										
	Conditions After stop of movement																										
	Unit 0.0001 mm/0.0001°																										
	Setting range ±99999999																										
M10	Command unit	Select a unit of command from the following table and set it. * micron system	<table border="1" style="margin-left: auto; margin-right: auto;"> <tr> <td></td><td>1000*</td><td>100*</td><td>10*</td><td>1*</td><td>0.1*</td></tr> <tr> <td>—</td><td>10000</td><td>1000</td><td>100</td><td>10</td><td>1</td></tr> <tr> <td>2° index</td><td>20000</td><td>2000</td><td>200</td><td>20</td><td>2</td></tr> <tr> <td>5° index</td><td>50000</td><td>5000</td><td>500</td><td>50</td><td>5</td></tr> </table>		1000*	100*	10*	1*	0.1*	—	10000	1000	100	10	1	2° index	20000	2000	200	20	2	5° index	50000	5000	500	50	5
	1000*	100*	10*	1*	0.1*																						
—	10000	1000	100	10	1																						
2° index	20000	2000	200	20	2																						
5° index	50000	5000	500	50	5																						
	Program type E																										
	Conditions At power on																										
	Unit —																										
	Setting range 1 to 50000																										
M11	Coding of address of axis	Register the address of each axis in hexadecimal numbers in ASCII code.	<table border="1" style="margin-left: auto; margin-right: auto;"> <tr> <td></td><td>X-axis</td><td>Y-axis</td><td>Z-axis</td><td>4th-axis</td><td>5th-axis</td><td>6th-axis</td></tr> <tr> <td>Address name</td><td>X</td><td>Y</td><td>Z</td><td>A</td><td>B</td><td>C</td></tr> <tr> <td>Set value</td><td>&58</td><td>&59</td><td>&5A</td><td>&41</td><td>&42</td><td>&43</td></tr> </table> <p style="text-align: center;">↑ ↑ ↑ Fixed value</p>		X-axis	Y-axis	Z-axis	4th-axis	5th-axis	6th-axis	Address name	X	Y	Z	A	B	C	Set value	&58	&59	&5A	&41	&42	&43			
	X-axis	Y-axis	Z-axis	4th-axis	5th-axis	6th-axis																					
Address name	X	Y	Z	A	B	C																					
Set value	&58	&59	&5A	&41	&42	&43																					
	Program type M, E																										
	Conditions At power on																										
	Unit —																										
	Setting range &0 to &7F																										

Classification	MACHINE	Display title	FEED VEL.
Address	Meaning	Description	
M12	Coding of incremental axis		Register the incremental axes respectively in hexadecimal numbers of the ASCII code.
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	&0 to &7F	
M13	Axis name (for display)		Specify the address name of the axes to be used for display, by the appropriate hexadecimal number of the ASCII code. For reverse display, set up the most significant bit.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	&0 to &FFFF	
M14	Shifting distance of the watchdog-less home position		When the watchdog-less home position is set, it will be shifted through the distance <If returning to the zero point is not yet executed>
			
			 <If returning to the zero point is executed>
			
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.0001°	
	Setting range	±99999999	

Classification	MACHINE	Display title	FEED VEL.
Address	Meaning	Description	
M15	Axis name (for axis name changing)	Specify the name of the address to be used for axis name changing (G110), by the appropriate hexadecimal number of the ASCII code.	
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	&0 to &7F	
M16	Zero-point shift amount/grid shift amount	<p>The distance from the grid point to the actual zero point that exists during zero-point return (reference point return) in the initial operation after power-on.</p>  <p>MPL527</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.0001°	
	Setting range	0 to 99999999	
M17	Axis control flag	<p>7 6 5 4 3 2 1 0</p> <ul style="list-style-type: none"> Unit of output from MCP to servo amplifier <ul style="list-style-type: none"> 0: Millimeter 1: Inch Direction of machine zero-point return <ul style="list-style-type: none"> 0: (+) direction 1: (-) direction Error correction schema with servo on <ul style="list-style-type: none"> 0: To correct with motor 1: To correct with counter display Type of axis <ul style="list-style-type: none"> 0: Linear 1: Rotational Rotational direction of servo motor (Note) <ul style="list-style-type: none"> (for movement in (+) direction) 0: CW 1: CCW If axis is removed: <ul style="list-style-type: none"> 0: Alarm 1: No alarm 	
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	MACHINE	Display title	FEED VEL.
Address	Meaning	Description	
M18 (bit 0)	Spindle C-axis	0: Motor type with frame 1: Built-in type	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
M18 (bit 2)	Machine zero-point position	0: Fixed point for zero-point return using dog 1: Position existing when power was turned on	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	
M18 (bit 3)	Virtual axis with dog	Set the axis to perform virtual dog-type zero-point return. 0: Invalid 1: Valid	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0, 1	
M18 (bit 4)	Dog-less axis	0: Invalid 1: Valid	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	

Classification	MACHINE	Display title	FEED VEL.																
Address	Meaning	Description																	
M18 (bit 5)	X-axis current position display in radius/diameter Program type M, E Conditions At power on Unit — Setting range 0, 1	0: Radius 1: Diameter																	
M18 (bit 6)	Automatic/manual simultaneous absolute-value updating Program type M, E Conditions At power on Unit — Setting range 0, 1	0: Invalid 1: Valid																	
M18 (bit 7)	Absolute-value detection Program type M, E Conditions At power on Unit — Setting range 0, 1	0: Invalid 1: Valid  <table border="1" data-bbox="738 1246 1341 1381"> <thead> <tr> <th colspan="2"></th> <th colspan="2">M18 bit 7</th> </tr> <tr> <th colspan="2"></th> <th>0</th> <th>1</th> </tr> </thead> <tbody> <tr> <td>SV17 bit 7</td> <td>0</td> <td>Dog type</td> <td>Simplified detection position</td> </tr> <tr> <td></td> <td>1</td> <td>Dog type</td> <td>Absolute detection position</td> </tr> </tbody> </table>			M18 bit 7				0	1	SV17 bit 7	0	Dog type	Simplified detection position		1	Dog type	Absolute detection position	
		M18 bit 7																	
		0	1																
SV17 bit 7	0	Dog type	Simplified detection position																
	1	Dog type	Absolute detection position																
M19 (bit 0)	Transmission of rapid feed rate value to SV69/SV70 Program type M, E Conditions At power on Unit — Setting range 0, 1	Set whether or not the value of parameter M1 (rapid feed rate) is transmitted to servo parameter at power-on. 0 : Transmit (same as before) 1 : Do not transmit																	

Classification	MACHINE	Display title	FEED VEL.
Address	Meaning	Description	
M19 (bit 3)	Select an in-position checking method Program type M, E Conditions At power on Unit — Setting range 0, 1	0 : NC 1 : Servo	
M19 (bit 4)	— Program type — Conditions — Unit — Setting range —	Parameter for system internal setting <u>Setting prohibited</u>	
M19 (bit 5)	Zero-point operation starting position check Program type M, E Conditions At power on Unit — Setting range 0, 1	0: No check 1: Alarm if returned from the dog	
M19 (bit 7)	Backlash scheme to be adopted for dog-type returning to zero point Program type M, E Conditions At power on Unit — Setting range 0, 1	0 : G01 1 : G0	

Classification	MACHINE	Display title	FEED VEL.
Address	Meaning	Description	
M20 (bit 0)	Rotational direction of the rotary axis (for dynamic offset II) Program type M, E Conditions At power on Unit Bit Setting range 0, 1	0: Forward 1: Reverse	
M20 (bit 1)	KOMET tool axis selection Program type M, E Conditions At power on Unit Bit Setting range 0, 1	0: Normal 1: KOMET tool axis	
M20 (bit 2)	Type of stopping the motion of feed and rapid traverse for the INTELLIGENT SAFETY SHIELD (ISS) Program type M, E Conditions At power on Unit Bit Setting range 0, 1	0: With the ISS being selected, the stop is delayed by about 140 ms than otherwise. Example : The stop position for combining the tailstock or steady rest will be shifted accordingly. 1: Independently of whether the ISS is selected or canceled, the axis movement will come to a stop in one and the same position. Example : Independently of the ISS, the stop for combining the tailstock or steady rest will occur in one and the same position. Changing the parameter setting from 0 to 1, therefore, must be followed by a readjustment of the combining position.)	
M20 (bit 6)	Switching between zero-point shift amount and grid shift amount Program type M, E Conditions At power on Unit — Setting range 0, 1	This parameter determines the use of parameter M16 . 0: Use parameter M16 for zero-point shift amount. 1: Use parameter M16 for grid shift amount.	

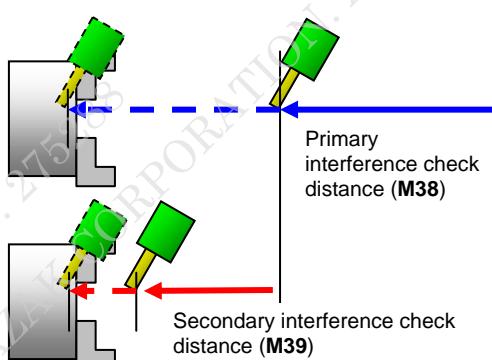
Classification	MACHINE	Display title	FEED VEL.
Address	Meaning		Description
M21	—	 	<p>0: Invalid 1: Valid</p> <p>Linear acceleration/ deceleration</p> <p>First-order lag</p> <p>Second-order lag</p> <p>Exponential acceleration/ linear deceleration</p> <p>Linear acceleration/ deceleration</p> <p>First-order lag</p> <p>Second-order lag</p> <p>Exponential acceleration/ linear deceleration</p> <p>Rapid-feed acceleration/ deceleration type</p> <p>Cutting-feed acceleration/ deceleration type</p>
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	<p>Time constants for each type of acceleration/deceleration control must be set using parameters N1 through N6.</p>
M22	—		<p>Deceleration time constant for rapid-feed exponential acceleration/linear deceleration</p> <p>0: Parameter N3</p> <p>1: Parameter N3 × 2</p> <p>Type of stroke-end stop</p> <p>00: Linear deceleration</p> <p>10: Position-loop stepped stop</p> <p>01: Speed-loop stepped stop</p> <p>11: Position-loop stepped stop</p>
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
M23 (bit 0)	Selection of NC control axes/PLC control axes		<p>0: NC control axes</p> <p>1: PLC control axes</p>
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0, 1	

Classification	MACHINE	Display title	FEED VEL.
Address	Meaning	Description	
M25 (Type A)	Illegal axis area upper limits (Type A)	<p>Set an access inhibition area in the inclined-axis control software limit area consisting of the X-axis and the Yt-axis.</p> <ul style="list-style-type: none"> Parameters M25 and M26 are valid only for the X-axis and the Yt-axis. If the X-axial and Yt-axial upper limits (M25) and lower limits (M26) are all zeros, the illegal axis area check function is invalid. Use the parameter BA126 to select Type A or B (BA126 bit 1). When the selection of whether to make this interference checking function valid or invalid is to be made according to the B-axis angle, specify the interference checking B-axis angle range. The interference checking function will be valid when the following conditions are satisfied: <ol style="list-style-type: none"> M26 (B-axis) \leq B-axis angle \leq M25 (B-axis) M25 (B-axis) = M26 (B-axis) = 0 	
Program type	M, E	<Type A (BA126 bit 1 = 0)>	
Conditions	After stop of movement		
Unit	0.0001 mm/0.0001°		
Setting range	± 99999999		
M26 (Type A)	Illegal axis area lower limits (Type A)		
Program type	M, E		
Conditions	After stop of movement		
Unit	0.0001 mm/0.0001°		
Setting range	± 99999999		

Classification	MACHINE	Display title	FEED VEL.
Address		Meaning	
M25 (Type B)		Illegal axis area upper limits (Type B)	
Program type M, E Conditions After stop of movement Unit 0.0001 mm/0.0001° Setting range ±99999999		Set an access inhibition area in the inclined-axis control software limit area consisting of the X-axis and the Yt-axis. <ul style="list-style-type: none"> Parameters M25 and M26 are valid only for the X-axis and the Yt-axis. If the X-axial and Yt-axial upper limits (M25) and lower limits (M26) are all zeros, the illegal axis area check function is invalid. Use the parameter BA126 to select Type A or B (BA126 bit 1). When the selection of whether to make this interference checking function valid or invalid is to be made according to the B-axis angle, specify the interference checking B-axis angle range. The interference checking function will be valid when the following conditions are satisfied: <ol style="list-style-type: none"> M26 (B-axis) ≤ B-axis angle ≤ M25 (B-axis) M25 (B-axis) = M26 (B-axis) = 0 <p><Type B (BA126 bit 1 = 1)></p>	
M26 (Type B)		Illegal axis area lower limits (Type B)	
Program type M, E Conditions After stop of movement Unit 0.0001 mm/0.0001° Setting range ±99999999		<p>Upper:</p> <p>Shaded area (A) in the above diagram, determined by parameters M8 (X), M9 (Yt), M25 (X), and M25 (Yt), is referred to as the illegal axis area. An alarm will result if an attempt is made to move the machine into the area.</p> <p>Lower:</p> <p>Shaded area (B) in the above diagram, determined by parameters M9 (X), M8 (Yt), M26 (X), and M26 (Yt), is referred to as the illegal axis area. An alarm will result if an attempt is made to move the machine into the area.</p>	
M27		Optimum acceleration control (Target speed) Program type M, E Conditions After stop of axis movement Unit mm/min Setting range 0 to 5000	
		Set the value that determines the maximum allowable value (Amax) of the acceleration that occurs between blocks. $A_{max} = \frac{M27}{N25} \times 60 \times 1000 \times \frac{100 - N26}{100} \text{ (mm/ms}^2\text{)}$ <p>Note :</p> <p>If M27 = 0, M27 is regarded as equal to M1 (rapid feed rate). If N25 = 0, N25 is regarded as equal to N1 (rapid feed time constant)</p>	

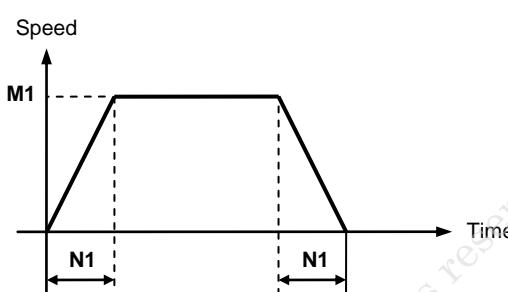
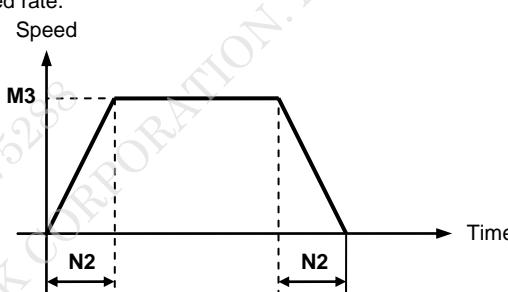
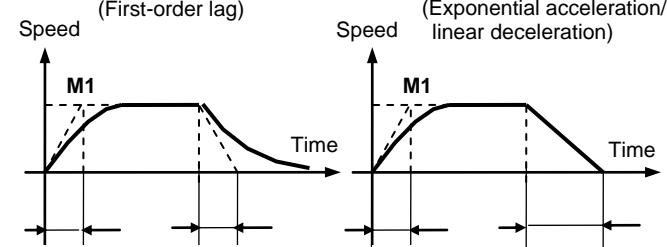
Classification	MACHINE	Display title	FEED VEL.
Address	Meaning	Description	
M29	Rapid feed clamping speed 1 for superposition control	<p>Set the rapid feed clamping speed 1 for superposition control. (Set value for the reference axis and superposition axis to be used during superposition control.)</p> <ul style="list-style-type: none"> When both the reference axis and superposition axis move at a rapid feed rate and in the same direction (see Note below), the rapid feed rate for superposition will be clamped in accordance with the setting of M30. Set M29 to M31 so that the following conditions are satisfied: <p>M29 (RA) + M31 (SA) ≤ M1 (SA) M31 (RA) + M29 (SA) ≤ M1 (SA) M30 (RA) + M30 (SA) ≤ M1 (SA) RA: reference axis M31 (RA) + M31 (SA) ≤ M1 (SA) SA: superposition axis</p> <p>Note : Whether the axes move in the same direction or in different directions is judged from the command and polarity (parameter N21, bit 6).</p>	
M30	Rapid feed clamping speed 2 for superposition control	<p>Set the rapid feed clamping speed for superposition control when both the reference axis and superposition axis move at a rapid feed rate and in the same direction (see the Note). (Set value for the reference axis and superposition axis to be used during superposition control.)</p> <ul style="list-style-type: none"> Set M29 to M31 so that the following conditions are satisfied: <p>M29 (RA) + M31 (SA) ≤ M1 (SA) M31 (RA) + M29 (SA) ≤ M1 (SA) M30 (RA) + M30 (SA) ≤ M1 (SA) RA: reference axis M31 (RA) + M31 (SA) ≤ M1 (SA) SA: superposition axis</p> <p>Note : Whether the axes move in the same direction or in different directions is judged from the command and polarity (parameter N21, bit 6).</p>	
M31	Cutting feed clamping speed for superposition control	<p>Set the cutting feed clamping speed for superposition control. (Set value for the reference axis and superposition axis to be used during superposition control.)</p> <ul style="list-style-type: none"> Set M29 to M31 so that the following conditions are satisfied: <p>M29 (RA) + M31 (SA) ≤ M1 (SA) M31 (RA) + M29 (SA) ≤ M1 (SA) M30 (RA) + M30 (SA) ≤ M1 (SA) RA: reference axis M31 (RA) + M31 (SA) ≤ M1 (SA) SA: superposition axis</p>	
M32	Safety speed (Safety supervisory mode 2)	Set the safety speed to be used in safety supervisory mode 2. If the operating speed of the axis exceeds the set value in safety supervisory mode 2, this will cause a safety supervisory alarm and result in an emergency stop.	
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	mm/min	
	Setting range	0 to 600000	

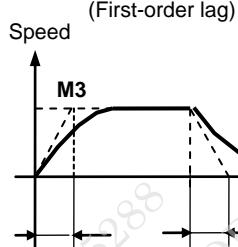
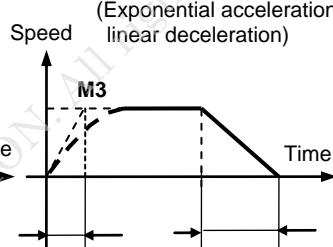
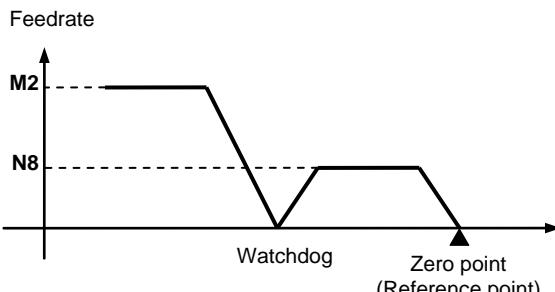
Classification	MACHINE	Display title	FEED VEL.
Address	Meaning		Description
M33	Safety speed (Safety supervisory mode 3)		Set the safety speed to be used in safety supervisory mode 3. If the operating speed of the axis exceeds the set value in safety supervisory mode 3, this will cause a safety supervisory alarm and result in an emergency stop. (Assign a value greater than that of parameter M32 to parameter M33 .)
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	mm/min / °/min	
	Setting range	0 to 99999999	
M34	Safety clamping speed (Safety supervisory mode 2)		Set the safety clamping speed to be used in safety supervisory mode 2. In safety supervisory mode 2 and while a safety clamping request is in effect, operation decelerates to the set speed. (Assign a value smaller than that of parameter M32 to parameter M34 .)
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	mm/min / °/min	
	Setting range	0 to 99999999	
M35	Safety clamping speed (Safety supervisory mode 3)		Set the safety clamping speed to be used in safety supervisory mode 3. In safety supervisory mode 3 and while a safety clamping request is in effect, operation decelerates to the set speed. (Assign a value smaller than that of parameter M33 to parameter M35 .)
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	mm/min / °/min	
	Setting range	0 to 99999999	
M36	Speed supervisory door selection (Safety supervisory mode)		Set to which door group the axis belongs in safety supervisory mode.
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	—	
	Setting range	0 to 99999999	

Classification	MACHINE	Display title	FEED VEL.
Address	Meaning		Description
M37	Safety clamping speed reduction judgment coefficient (Safety supervisory mode)		A speed clamping signal is output after the axis has decelerated to the clamping speed. This parameter specifies to what additional percentage of the safety clamping speed the axis is to be decelerated as the output timing of the speed clamping signal. If 0 is set, the speed clamping signal will be output when the axis decelerates to a 10% additional speed (i.e., 110% of the safety clamping speed).
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	%	
	Setting range	0 to 99999999	
M38 M39	Interference check distances for INTELLIGENT SAFETY SHIELD during manual operation M38: Primary check distance M39: Secondary check distance		<p>Set the machine interference detection distances for the INTELLIGENT SAFETY SHIELD. Set data so that the primary check distance is longer than the secondary check distance.</p>  <p>Note : The same value as that of the primary parallel driving control axis should be assigned to the secondary axis.</p>
M40 M41	Ram spindle stored stroke end M40: Upper limit M41: Lower limit		For the machine specifications with a ram spindle, specify the maximum operating range for the ram spindle selection. When ram spindle is selected, the range specified in these parameters becomes the maximum operating range, instead of the manufacturer software limits specified in M08/M09 .
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	0.0001 mm/0.0001°	
	Setting range	±99999999	

Classification	MACHINE	Display title	FEED VEL.
Address	Meaning		Description
M42	Interference check distance for INTELLIGENT SAFETY SHIELD during automatic operation		Set the machine interference check distance for INTELLIGENT SAFETY SHIELD during automatic operation Note : The same value as that of the primary parallel driving control axis should be assigned to the secondary axis.
	Program type	M, E	
	Conditions	At power on	
	Unit	0.0001 mm/0.0001°	
	Setting range	0 to 99999999	
M45	Selection of a door for signal input on the drive side (servo) in safety supervisory mode		Set which door signal is input for the axis with door signal input on the drive side (servo) in safety supervisory mode. 1: Door 1 signal 2: Door 2 signal 4: Door 3 signal 8: Door 4 signal
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 255	

2-3-13 Machine parameter TIME CONST. (N)

Classification	MACHINE	Display title	TIME CONST.
Address	Meaning		Description
N1	Rapid-feed time constant (linear acceleration/deceleration)		<p>Set the time constant to obtain linear acceleration/deceleration of the rapid feed rate.</p>  <p>Note : This parameter is valid only when bit 0 of M21 is 1.</p> <p style="text-align: right;">MPL523</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	ms	
	Setting range	4 to 1800	
N2	Cutting-feed time constant (linear acceleration/deceleration)		<p>Set the time constant to obtain linear acceleration/deceleration of the cutting feed rate.</p>  <p>Note : This parameter is valid only when bit 4 of M21 is 1.</p> <p style="text-align: right;">MPL523</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	ms	
	Setting range	4 to 1800	
N3	Rapid-feed time constant (First-order lag)		<p>First-order lag time constant for rapid-feed acceleration/deceleration (First-order lag)</p>  <p>Note : This parameter is valid only when either bit 1, 2 or 3 of M21 is 1.</p> <p style="text-align: right;">MPL524</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	ms	
	Setting range	4 to 5000	

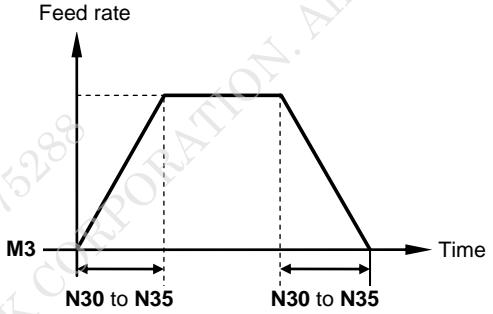
Classification	MACHINE	Display title	TIME CONST.
Address	Meaning	Description	
N4	Time constant for post-interpolation rapid feed acceleration/deceleration filter	Set the time constant for the filter that further smoothes the speed signal waveform during rapid feed (G0) operation with shape correction off and during rapid feed operation in manual operation mode. The filter will be invalid if 0 is set.	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	ms	
	Setting range	0 to 455	
N5	Cutting-feed time constant (First-order lag)	First-order lag time constant for cutting-feed acceleration/ deceleration (First-order lag)  Speed M3 N5 Time Exponential acceleration/ linear deceleration  Speed M3 N5 N5 x 2 Time Note : This parameter is valid only when either bit 5, 6 or 7 of M21 is 1. MPL524	
	Program type	M, E	
	Conditions	Immediate	
	Unit	ms	
	Setting range	0 to 5000	
N7	OT time	During external deceleration, the position loop is disconnected for the time interval set using this parameter and, as a result, the speed becomes zero.	
	Program type	M, E	
	Conditions	Immediate	
	Unit	ms	
	Setting range	1 to 32767	
N8	Creeping speed during initial zero-point return	The feed rate at which each axis is moved back to the zero-point (reference point) after the zero-point watchdog LS (limit switch) has turned on in the initial operation after power-on.  Feedrate M2 N8 Watchdog Zero point (Reference point) MPL525	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	mm/min (°/min)	
	Setting range	1 to 60000	

Classification	MACHINE	Display title	TIME CONST.
N9	Amount of grid ignorance during initial zero-point return	The spacing at which the grid point is ignored during zero-point return (reference point return in the initial operation after power-on) after the zero-point watchdog LS (limit switch) is turned off. With this parameter, dispersion in position deviations of the zero point can be avoided.	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm (0.001°)	
	Setting range	0 to 65535	
N10	Grid interval	Set the grid interval of the detector. Usually, set the same value as the ball screw pitch. However, set the grid interval of the detector if the grid interval differs from the pitch.	
	Program type	M, E	
	Conditions	At power on	
	Unit	mm (0.001°)	
	Setting range	0 to 32767	
N12	—	Machine dependent specific value	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
N13	—	Machine dependent specific value	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

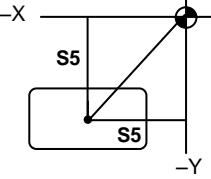
Classification	MACHINE	Display title	TIME CONST.
Address	Meaning	Description	
N15	— Program type Conditions Unit Setting range	Machine dependent specific value	
N16	— Program type Conditions Unit Setting range	Machine dependent specific value	
N17	Servo amplifier channel number Program type Conditions Unit Setting range	Specify the channel number of the servo amplifier. 1: 1 ch 2: 2 ch 3: 3 ch	
N18	Servo amplifier rotary switch number Program type Conditions Unit Setting range	Specify the rotary switch number of the servo amplifier. 0: SW0 1: SW1 2: SW2 3: SW3 4: SW4 5: SW5 6: SW6	

Classification	MACHINE	Display title	TIME CONST.
Address	Meaning	Description	
N19	Axis system number Program type M, E Conditions At power on Unit — Setting range 0 to 3	Set the system number of the axes. 0: System 1 1: System 2 2: System 3 3: System 4	
N21	— Program type M, E Conditions At power on Unit Bit Setting range Binary, eight digits	<p>Linear-type rotational axis (Note) 0: Rotary type 1: Linear type</p> <p>Rotational axis shortcut 0: Invalid 1: Valid</p> <p>Fixed value (0)</p> <p>Fixed value</p> <p>Reference axis for superposition control</p> <p>Superposition axis for superposition control</p> <p>Relative polarity of control axis</p> <p>Current position immediately after cross machining 0: No re-acquisition 1: Re-acquisition</p> <p>Note : When M18 bit 0 = 1 (type of C-axis: built-in type), if N21 bit 0 is set to 1 (linear type rotational axis), SA46 bit 2 must be set to 0 (C-axis position control changeover type: after return to zero point).</p>	
N22 (bit 0)	Axis selection for positioning during automatic operation or virtual machining with 3D interference check Program type M, E Conditions At power on Unit Bit Setting range 0, 1	0: Positioning 1: No positioning	

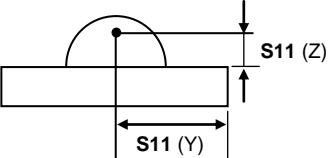
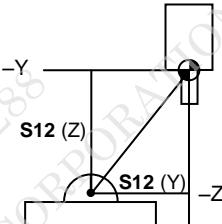
Classification	MACHINE	Display title	TIME CONST.
Address	Meaning		Description
N25	Time constant for deceleration rate calculation		<p>Set the maximum allowable value, Amax, for the acceleration occurring between blocks.</p> $A_{\text{max}} = \frac{M_{27}}{N_{25}} \times 60 \times 1000 \times \frac{100 - N_{26}}{100} \text{ (mm/ms}^2\text{)}$ <p>Note :</p> <p>If M27 = 0, M27 is regarded as equal to M1 (rapid feed rate). If N25 = 0, N25 is regarded as equal to N1 (rapid feed time constant)</p>
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	ms	
	Setting range	0 to 5000	
N26	Accuracy coefficient for deceleration rate calculation		
	Program type	M, E	
	Conditions	After stop of axis movement	
	Unit	—	
	Setting range	±32768	
N27	Rapid feed time constant for superposition		<p>Set the rapid feed (linear acceleration/deceleration) time constant for superposition control.</p> <p>Note : Set the same value for all axes.</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	ms	
	Setting range	4 to 1800	
N28	Cutting feed time constant for superposition		<p>Set the cutting feed (linear acceleration/deceleration) time constant for superposition control.</p> <p>Note : Set the same value for all axes.</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	ms	
	Setting range	4 to 1800	

Classification	MACHINE	Display title	TIME CONST.
Address	Meaning	Description	
N29	Time constant for shape correction rapid feed acceleration/deceleration filter	<p>Use of this parameter allows further smoothening of the speed signal waveform during rapid feed with shape correction on. The filter will be invalid if 0 is set.</p>	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	ms	
	Setting range	0 to 455	
N30 to N35	Cutting feed time constant for time constant changeover M-code command N30: for M881 command N31: for M882 command N32: for M883 command N33: for M884 command N34: for M885 command N35: for M886 command	<p>Set the cutting feed time constant for a time constant changeover M-code command.</p> 	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	ms	
	Setting range	0 to 1800	

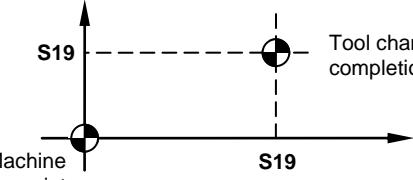
2-3-14 Machine parameter ANOTHER (S)

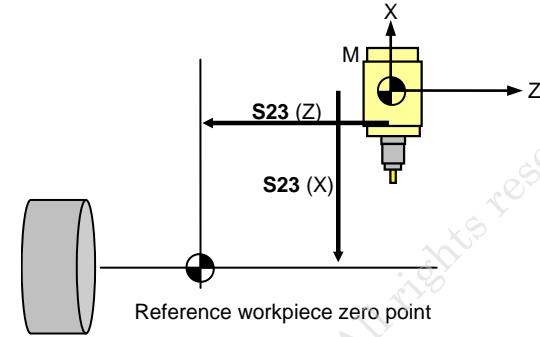
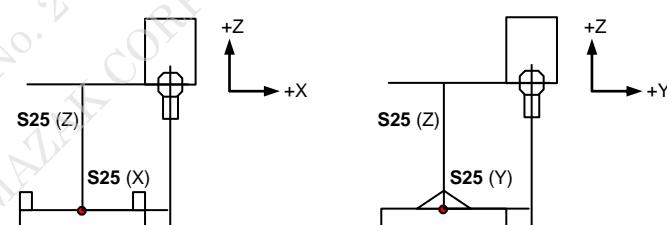
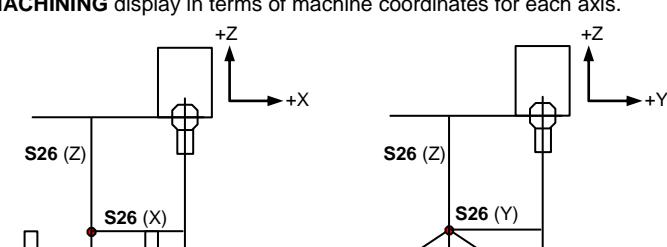
Classification	MACHINE	Display title	ANOTHER
Address	Meaning		Description
S3	Feed forward gain for the MAZAK Precision Rapid Boring Tornado Option		Set for each axis the feed forward gain for acceleration/deceleration before interpolation for the MAZAK Precision Rapid Boring Tornado Option.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.1%	
	Setting range	0 to 1000	
	Feed forward gain		Set for each axis the feed forward gain for acceleration/deceleration before interpolation.
S4	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
	Rotational center of the table		Set the position of the rotational center of the table in the machine coordinates system.
S5			 <p>Note :</p> <ol style="list-style-type: none"> 1. Z-axis data are not required for a machine with a tilting table. 2. This parameter is used to execute the automatic program origin calculation function (option) for a machine with a tilting table.
	Program type	M, E	(For INTEGREX series)
	Conditions	At power on	(For dynamic offset)
	Unit	0.0001 mm/0.00001 in	(For HV machining)
	Setting range	0 to ±99999999	(For machines equipped with a tilting table)

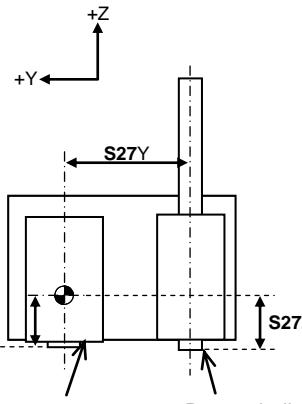
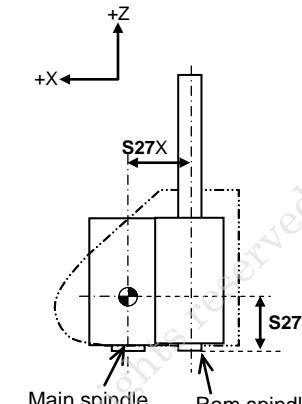
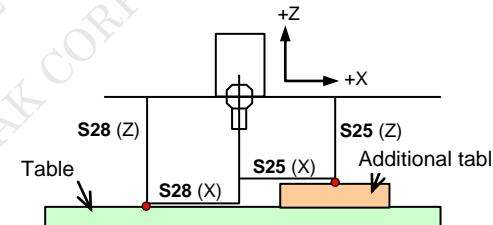
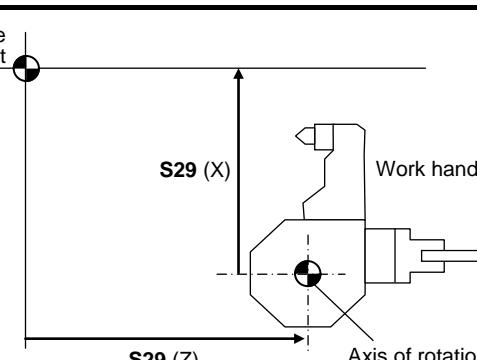
Classification	MACHINE	Display title	ANOTHER
Address	Meaning	Description	
S6	Absolute position detection parameter Program type M, E Conditions At power on Unit 0.0001 mm/0.00001 in/ 0.0001° Setting range 0 to 99999999	When movement is beyond the length set by this parameter during the power off, it activates the alarm mode.	
S7	Upper limit (on Z-axis) of machining range for table rotating machining I Program type M, E Conditions At power on Unit 0.0001 mm/0.00001 in Setting range 0 to ±99999999	This parameter specifies the range of rotating machining for the table rotating machining I (X-B machining). Set the upper limit (on Z-axis) of the machining range in the machine coordinates system. The machine recognizes that it is prohibited to move beyond this limit in the negative direction. (For HV machining)	
S8	Feed-forward gain G00 Program type M, E Conditions Immediate Unit % Setting range 0 to 100	Specify the pre-interpolation acceleration/deceleration feed-forward gain levels for each axis.	
S10	Axis of rotation of the tilting table Program type M, E Conditions Immediate Unit 0.0001 mm Setting range 0 to ±99999999	Specify the axis of rotation of the tilting table in machine coordinates (Y and Z). This parameter is used for the software travel limit function. Note : X-axial data are not required. (For machines equipped with a tilting table)	

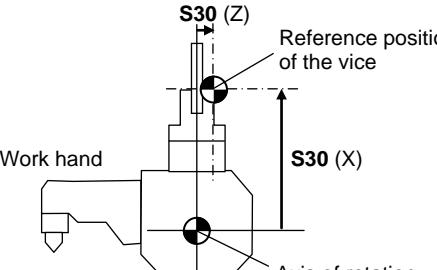
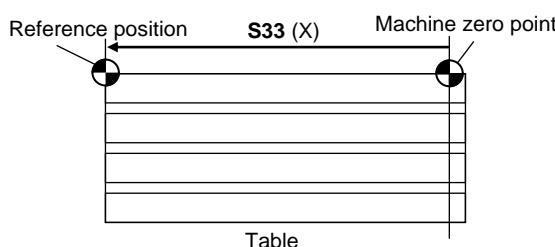
Classification	MACHINE	Display title	ANOTHER
Address	Meaning	Description	
S11	<p>Corner position of the tilting table</p> <p>Program type M, E</p> <p>Conditions Immediate</p> <p>Unit 0.0001 mm</p> <p>Setting range 0 to 99999999</p>	<p>Specify the corner position of the tilting table in (Y- and Z-axial) distances from its axis of rotation.</p> <p>This parameter is used for the software travel limit function.</p> <p>Note : X-axial data are not required.</p>  <p>(For machines equipped with a tilting table)</p>	
S12	<p>Axis of rotation of the tilting table (Used for the automatic program origin calculation function)</p> <p>Program type M, E</p> <p>Conditions Immediate</p> <p>Unit 0.001 mm/0.0001 in</p> <p>Setting range 0 to ±99999999</p>	<p>Specify the axis of rotation of the tilting table in machine coordinates (Y and Z). Use this parameter to execute the automatic program origin calculation function (option).</p> <p>Measure and enter data for respective machines.</p> <p>Note : X-axial data are not required.</p>  <p>(For machines equipped with a tilting table)</p>	
S13	<p>G00 in-position width</p> <p>Program type M, E</p> <p>Conditions At power on</p> <p>Unit 0.001 mm</p> <p>Setting range 0 to 32767</p>	<p>Set the in-position width for G00. The in-position check for G00 is effective when the parameter K95 bit 7 is 1 and K96 bit 0 is 0.</p> <p>For utilizing the in-position width of G00, set the in-position width of the servo parameter to 0 to avoid trouble.</p> <p>Note : This parameter is valid only when bit 3 of M19 is 0.</p>	
S14	<p>G01 in-position width</p> <p>Program type M, E</p> <p>Conditions At power on</p> <p>Unit 0.001 mm</p> <p>Setting range 0 to 32767</p>	<p>Set the in-position width for G01. The in-position check for G01 is effective when one of G09 (exact stop check), G61 (exact stop check mode) and the error detection is selected with the parameter K95 bit 7 set to 1.</p> <p>For utilizing the in-position width for G01, set the in-position width of the servo parameter to 0 to avoid trouble.</p> <p>Note : This parameter is valid only when bit 3 of M19 is 0.</p>	

Classification	MACHINE	Display title	ANOTHER
Address	Meaning		Description
S15	 — Program type — Conditions — Unit — Setting range —		Fixed value
S16	 Unbalanced axis torque offset Program type M Conditions At power on Unit — Setting range —		Set automatically after estimation of the characteristics.
S17	 Torque limit buffer reduction ratio 1 Program type M, E Conditions Immediate Unit % Setting range 0 to 200		Specify torque limit buffer reduction ratio 1.
S18	 Torque limit buffer reduction ratio 2 Program type M, E Conditions Immediate Unit % Setting range 0 to 200		Specify torque limit buffer reduction ratio 2.

Classification	MACHINE	Display title	ANOTHER								
Address	Meaning	Description									
S19	<p>Tool change completion position of the long boring bar end tool</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001 in 0.0001°</td></tr> <tr><td>Setting range</td><td>±99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.0001 mm/0.00001 in 0.0001°	Setting range	±99999999	<p>Specify the tool change completion position of the long boring bar end tool by the corresponding machine coordinates.</p>  <p>MPL521</p>	
Program type	M, E										
Conditions	Immediate										
Unit	0.0001 mm/0.00001 in 0.0001°										
Setting range	±99999999										
S20	<p>APC completion position</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001°</td></tr> <tr><td>Setting range</td><td>±99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.0001 mm/0.00001°	Setting range	±99999999	Set the completion position of automatic pallet change operation.	
Program type	M, E										
Conditions	Immediate										
Unit	0.0001 mm/0.00001°										
Setting range	±99999999										
S21	<p>Completion position of gantry robot operation</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001°</td></tr> <tr><td>Setting range</td><td>±99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.0001 mm/0.00001°	Setting range	±99999999	Set the completion position of gantry robot operation.	
Program type	M, E										
Conditions	Immediate										
Unit	0.0001 mm/0.00001°										
Setting range	±99999999										
S22	<p>Cutting feed clamping speed during tool tip point control</p> <table border="1"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>mm/min (%/min)</td></tr> <tr><td>Setting range</td><td>1 to 200000</td></tr> </table>	Program type	E	Conditions	After stop of movement	Unit	mm/min (%/min)	Setting range	1 to 200000	<p>Specify the critical cutting feed rate to be used during tool tip point control. During tool tip point control, speed is clamped in accordance with parameter S22 or M3 (critical cutting feed rate), whichever is the smaller.</p>	
Program type	E										
Conditions	After stop of movement										
Unit	mm/min (%/min)										
Setting range	1 to 200000										

Classification	MACHINE	Display title	ANOTHER
Address	Meaning	Description	
S23	Reference workpiece zero point	<p>Set the position of a reference workpiece zero point for each axis in the machine coordinate system.</p> <p>Set the position of a reference workpiece zero point relating to the turret also in the machine coordinate system.</p> <p>For machining centers, this parameter is invalid.</p> <p>* Tool set value is determined based on this parameter.</p>  <p>Reference workpiece zero point</p>	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	0.0001 mm/0.00001 in	
	Setting range	± 99999999	
S25	Central position on the upper surface of the tilt table (for machines with a tilt table) Central position on the table upper surface (for other machining centers)	<p>Set the central position of the tilt table upper surface on the VIRTUAL MACHINING display in terms of machine coordinates for each axis. (For machines with a tilt table)</p> <p>Set the central position of the table upper surface on the VIRTUAL MACHINING display in terms of machine coordinates for each axis. (For other machining centers)</p> 	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 in	
	Setting range	± 99999999	
S26	Central position of tilt table rotation	Set the position of the tilt table rotational center on the VIRTUAL MACHINING display in terms of machine coordinates for each axis.	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001 mm/0.00001 in	
	Setting range	± 99999999	 <p>(Special function for machines with a tilt table)</p>

Classification	MACHINE	Display title	ANOTHER	
Address	Meaning	Description		
S27	Ram spindle offset amount		For the machine specifications with a ram spindle, specify the offsets between the main spindle and the ram spindle for each axis.	
				
				
	<p>(Special function for machines with a ram spindle)</p>			
	Program type	E		
S28	Conditions	Immediate		
	Unit	0.0001 mm		
	Setting range	± 99999999		
	Center of the workpiece to be directly mounted on the table of the machines equipped with an additional table		For graphics on the VIRTUAL MACHINING display, set with machine coordinates of the rectangular axes, the center of the workpiece to be directly mounted on the table of the machines equipped with an additional table.	
				
S29	Note :			
	This parameter is only used when multiple material pieces can be arranged on the display in question (F35 bit 0 = 1). Use parameter S25 to set the center of the additional table's upper surface.			
	Position of the axis of rotation of the work hand in relation to the machine zero point			
	Program type	M, E		
	Conditions	Immediate		
	Unit	0.0001 mm/0.00001 in		
	Setting range	± 99999999	(Only for machines with a workpiece handling device)	

Classification	MACHINE	Display title	ANOTHER								
Address	Meaning	Description									
S30	<p>Distance from the axis of rotation of the work hand to the reference position of the vice</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001 in</td></tr> <tr><td>Setting range</td><td>± 99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.0001 mm/0.00001 in	Setting range	± 99999999	<p>Set the distance from the axis of rotation of the work hand to the reference position of the vice for the work hand during workpiece transfer.</p>  <p>(Only for machines with a workpiece handling device)</p>	
Program type	M, E										
Conditions	Immediate										
Unit	0.0001 mm/0.00001 in										
Setting range	± 99999999										
S31	<p>Specification of the machine coordinates of the floating reference point</p> <table border="1"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001 in</td></tr> <tr><td>Setting range</td><td>± 99999999</td></tr> </table>	Program type	E	Conditions	Immediate	Unit	0.0001 mm/0.00001 in	Setting range	± 99999999	<p>Specify the machine coordinates of the floating reference point.</p> <p>Example : G30.1X10.Z10.;</p> <p>Entry of the above command returns the tool in rapid feed mode through the intermediate point (X10, Z10) to the floating reference point specified in this parameter.</p>	
Program type	E										
Conditions	Immediate										
Unit	0.0001 mm/0.00001 in										
Setting range	± 99999999										
S32	<p>Rapid traverse speed for the mode of tool tip point control</p> <table border="1"> <tr><td>Program type</td><td>E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>mm/min (%/min)</td></tr> <tr><td>Setting range</td><td>0 to 200000</td></tr> </table>	Program type	E	Conditions	Immediate	Unit	mm/min (%/min)	Setting range	0 to 200000	<p>The general setting for rapid traverse speed in parameter M1 will be used also for the mode of tool tip point control if it is lower than the setting in parameter S32.</p> <p>Note : Set zero (0) in S32 if it is desirable to use the general setting in parameter M1 for the rapid traverse in question.</p>	
Program type	E										
Conditions	Immediate										
Unit	mm/min (%/min)										
Setting range	0 to 200000										
S33	<p>Reference position X for the TEACH function, to be used for setting the LENGTH B item of an angular tool (For machines of five-surface machining by angular tools)</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.0001 mm</td></tr> <tr><td>Setting range</td><td>—</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.0001 mm	Setting range	—	<p>Set the machine coordinate of the table edge on the negative side of the X-axis.</p> 	
Program type	M, E										
Conditions	Immediate										
Unit	0.0001 mm										
Setting range	—										

Classification	MACHINE	Display title	ANOTHER								
Address		Meaning	Description								
S35		Rotational center of the second table	<p>Set the position of the rotational center of the second table in the machine coordinate system when table selection is valid. (That of the first table to be specified in parameter S5)</p> <p>Table having a horizontally rotational axis</p> <p>Table having a vertically rotational axis</p> <p>Note :</p> <p>Among the rotary axes (parameter M17 bit 4 = 1) excluding the rotary head, the first axis is regarded as the first rotary table and the second axis the second rotary table.</p>								
<table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="padding: 2px;">Program type</td> <td style="padding: 2px;">M, E</td> </tr> <tr> <td style="padding: 2px;">Conditions</td> <td style="padding: 2px;">At power on</td> </tr> <tr> <td style="padding: 2px;">Unit</td> <td style="padding: 2px;">0.0001 mm/0.00001 in</td> </tr> <tr> <td style="padding: 2px;">Setting range</td> <td style="padding: 2px;">0 to ±99999999</td> </tr> </table>				Program type	M, E	Conditions	At power on	Unit	0.0001 mm/0.00001 in	Setting range	0 to ±99999999
Program type	M, E										
Conditions	At power on										
Unit	0.0001 mm/0.00001 in										
Setting range	0 to ±99999999										
S38		Time-out period for releasing the emergency stop status during powering on	<p>This parameter allows you to make the preset time for canceling the emergency stop status at powering on (hotline disconnection time) variable.</p> <p>0: 1000 ms 1 to 7: 8 ms 8 to 3000: The time-out period is set according to the setting. Beyond 3000: 3000 ms</p> <p>Note :</p> <ol style="list-style-type: none"> 1. Set the value for the first axis (X-axis). 2. Setting a negative value invalidates this parameter and the hotline is disconnected for 7.1 ms. 3. When this parameter is valid, the time taken to cancel the emergency stop status is output to register R71. When the ready lamp turning on process has failed due to a communication error, "0" is output to the register R71. 								
<table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="padding: 2px;">Program type</td> <td style="padding: 2px;">M, E</td> </tr> <tr> <td style="padding: 2px;">Conditions</td> <td style="padding: 2px;">At power on</td> </tr> <tr> <td style="padding: 2px;">Unit</td> <td style="padding: 2px;">ms</td> </tr> <tr> <td style="padding: 2px;">Setting range</td> <td style="padding: 2px;">0 to 3000</td> </tr> </table>				Program type	M, E	Conditions	At power on	Unit	ms	Setting range	0 to 3000
Program type	M, E										
Conditions	At power on										
Unit	ms										
Setting range	0 to 3000										

2-3-15 Machine parameter SPINDLE (SA)

Classification	MACHINE	Display title	SPINDLE																													
Address	Meaning	Description																														
SA1 to SA8	Maximum spindle speed in each speed range (range 1 to 8)	Number of revolutions per minute of the spindle in each speed range																														
		<table border="1" style="margin-left: auto; margin-right: auto;"> <thead> <tr> <th rowspan="2">Address</th> <th colspan="4">Maximum number of speed ranges</th> </tr> <tr> <th>1</th> <th>2</th> <th>3</th> <th>4</th> </tr> </thead> <tbody> <tr> <td>SA1</td> <td>O</td> <td>L</td> <td>L</td> <td>L</td> </tr> <tr> <td>SA2</td> <td>Invalid</td> <td>H</td> <td>M</td> <td>ML</td> </tr> <tr> <td>SA3</td> <td>Invalid</td> <td>Invalid</td> <td>H</td> <td>MH</td> </tr> <tr> <td>SA4</td> <td>Invalid</td> <td>Invalid</td> <td>Invalid</td> <td>H</td> </tr> </tbody> </table>		Address	Maximum number of speed ranges				1	2	3	4	SA1	O	L	L	L	SA2	Invalid	H	M	ML	SA3	Invalid	Invalid	H	MH	SA4	Invalid	Invalid	Invalid	H
Address	Maximum number of speed ranges																															
	1	2	3	4																												
SA1	O	L	L	L																												
SA2	Invalid	H	M	ML																												
SA3	Invalid	Invalid	H	MH																												
SA4	Invalid	Invalid	Invalid	H																												
Example :																																
<p>Output voltage</p>																																
Program type M, E																																
Conditions At power on																																
Unit min⁻¹																																
Setting range 0 to 99999																																
Constants for calculating each gear speed of the spindle																																
SA9 to SA16	Constants for calculating each gear speed of the spindle (range 1 to 8)	<table border="1" style="margin-left: auto; margin-right: auto;"> <thead> <tr> <th rowspan="2">Address</th> <th colspan="4">Maximum number of speed ranges</th> </tr> <tr> <th>1</th> <th>2</th> <th>3</th> <th>4</th> </tr> </thead> <tbody> <tr> <td>SA9</td> <td>O</td> <td>L</td> <td>L</td> <td>L</td> </tr> <tr> <td>SA10</td> <td>Invalid</td> <td>H</td> <td>M</td> <td>ML</td> </tr> <tr> <td>SA11</td> <td>Invalid</td> <td>Invalid</td> <td>H</td> <td>MH</td> </tr> <tr> <td>SA12</td> <td>Invalid</td> <td>Invalid</td> <td>Invalid</td> <td>H</td> </tr> </tbody> </table>		Address	Maximum number of speed ranges				1	2	3	4	SA9	O	L	L	L	SA10	Invalid	H	M	ML	SA11	Invalid	Invalid	H	MH	SA12	Invalid	Invalid	Invalid	H
Address	Maximum number of speed ranges																															
	1	2	3	4																												
SA9	O	L	L	L																												
SA10	Invalid	H	M	ML																												
SA11	Invalid	Invalid	H	MH																												
SA12	Invalid	Invalid	Invalid	H																												
Program type M, E																																
Conditions At power on																																
Unit min⁻¹																																
Setting range 0 to 99999																																
(⇒ SA1, SA2, SA3, SA4)																																
The maximum number of revolutions per minute of the spindle in each speed range during a tapping cycle																																
<table border="1" style="margin-left: auto; margin-right: auto;"> <thead> <tr> <th rowspan="2">Address</th> <th colspan="4">Maximum number of speed ranges</th> </tr> <tr> <th>1</th> <th>2</th> <th>3</th> <th>4</th> </tr> </thead> <tbody> <tr> <td>SA17</td> <td>O</td> <td>L</td> <td>L</td> <td>L</td> </tr> <tr> <td>SA18</td> <td>Invalid</td> <td>H</td> <td>M</td> <td>ML</td> </tr> <tr> <td>SA19</td> <td>Invalid</td> <td>Invalid</td> <td>H</td> <td>MH</td> </tr> <tr> <td>SA20</td> <td>Invalid</td> <td>Invalid</td> <td>Invalid</td> <td>H</td> </tr> </tbody> </table>		Address	Maximum number of speed ranges				1	2	3	4	SA17	O	L	L	L	SA18	Invalid	H	M	ML	SA19	Invalid	Invalid	H	MH	SA20	Invalid	Invalid	Invalid	H		
Address	Maximum number of speed ranges																															
	1	2	3	4																												
SA17	O	L	L	L																												
SA18	Invalid	H	M	ML																												
SA19	Invalid	Invalid	H	MH																												
SA20	Invalid	Invalid	Invalid	H																												
Program type M, E																																
Conditions At power on																																
Unit min⁻¹																																
Setting range 0 to 99999																																

Classification	MACHINE	Display title	SPINDLE																													
Address	Meaning	Description																														
SA25 to SA28	Spindle speed during gear shifting (range 1 to 4)		The number of revolutions per minute of the spindle during shifting of gears thru the various ranges																													
	Program type	M, E	<table border="1"> <thead> <tr> <th>Address</th><th colspan="4">Maximum number of speed range</th></tr> <tr> <th></th><th>1</th><th>2</th><th>3</th><th>4</th></tr> </thead> <tbody> <tr> <td>SA25</td><td>O</td><td>L</td><td>L</td><td>L</td></tr> <tr> <td>SA26</td><td>Invalid</td><td>H</td><td>M</td><td>ML</td></tr> <tr> <td>SA27</td><td>Invalid</td><td>Invalid</td><td>H</td><td>MH</td></tr> <tr> <td>SA28</td><td>Invalid</td><td>Invalid</td><td>Invalid</td><td>H</td></tr> </tbody> </table>	Address	Maximum number of speed range					1	2	3	4	SA25	O	L	L	L	SA26	Invalid	H	M	ML	SA27	Invalid	Invalid	H	MH	SA28	Invalid	Invalid	Invalid
Address	Maximum number of speed range																															
	1	2	3	4																												
SA25	O	L	L	L																												
SA26	Invalid	H	M	ML																												
SA27	Invalid	Invalid	H	MH																												
SA28	Invalid	Invalid	Invalid	H																												
Conditions	At power on																															
Unit	min ⁻¹																															
Setting range	0 to 32767																															
SA29	Load meter reference output scaling factor (high-speed coil)		Scaling factor (for high-speed coil) with respect to the reference output of the load meter																													
	Program type	M, E																														
	Conditions	Immediate																														
	Unit	%																														
	Setting range	0 to 100																														
SA30	Load meter reference output base rotational speed (high-speed coil)		Base rotational speed (for high-speed coil) for the output changed by the parameter SA29 with respect to the reference output of the load meter																													
	Program type	M, E																														
	Conditions	Immediate																														
	Unit	min ⁻¹																														
	Setting range	0 to 32767																														
SA31	Load meter reference output scaling factor (low-speed coil)		Scaling factor (for low-speed coil) with respect to the reference output of the load meter																													
	Program type	M, E																														
	Conditions	Immediate																														
	Unit	%																														
	Setting range	0 to 100																														

Classification	MACHINE	Display title	SPINDLE
Address	Meaning	Description	
SA32	Load meter reference output base rotational speed (low-speed coil)	Base rotational speed (for low-speed coil) for the output changed by the parameter SA31 with respect to the reference output of the load meter	<p>Output (kW)</p> <p>Speed (min^{-1})</p> <p>Output diagram at short-time rating</p> <p>Output diagram at continuous rating</p> <p>$\mathbf{SA31} = P_1/P_0 \times 100$</p> <p>$\mathbf{SA32} = N_1$</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	min^{-1}	
	Setting range	0 to 32767	
SA33 to SA40	Acceleration/deceleration time constant for the synchronous tapping cycle (range 1 to 8)	Linear acceleration/deceleration time constant for the synchronous tapping cycle	<p>SA33: Range 1</p> <p>SA34: Range 2</p> <p>SA35: Range 3</p> <p>SA36: Range 4</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	ms	
	Setting range	0 to 1800	
SA41	Spindle orientating speed	Specify the spindle orientating speed.	
	Program type	M, E	
	Conditions	At power on	
	Unit	min^{-1}	
	Setting range	0 to 32767	
SA42	Minimum spindle speed	Specify the minimum spindle speed.	
	Program type	M, E	
	Conditions	At power on	
	Unit	min^{-1}	
	Setting range	0 to 32767	

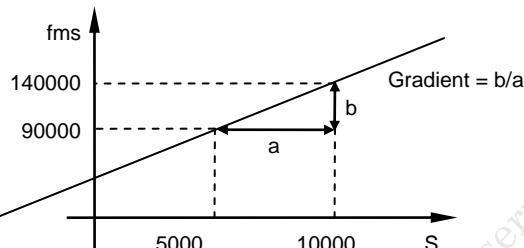
Classification	MACHINE	Display title	SPINDLE
Address	Meaning	Description	
SA43	Channel number for the spindle amplifier		Specify the channel number for the spindle amplifier. 1: 1 ch (the setting used when connection to SV1 of the HR353 is established) 2: 2 ch (the setting used when connection to SV3 of the HR353 is established)
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 2	
SA44	Spindle amplifier rotary switch number		Specify the rotary switch number of the spindle amplifier. 0: SW0 1: SW1 2: SW2 3: SW3 4: SW4 5: SW5 6: SW6
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 6	
SA45 (bit 0)	Spindle speed range changing method, in relation to switching the torque factors for auto-pecking of the cutting load detection type		0: As specified by bit 1 1: L/H coil sw. + Mecha. gear shift * For the setting of SA45 bit 0 = 1, set the rated torque, viscous & coulombic friction coefficients of the spindle motor in the relevant parameters as tabulated below:
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
SA45 (bit 1)	Spindle speed range changing method, in relation to switching the torque factors for auto-pecking of the cutting load detection type 2		0: L/H coil switching scheme 1: Mecha-gear scheme (Only when 0 is set up at bit 0) * Bit 1 is valid only when bit 0 = 0.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	

Classification	MACHINE	Display title	SPINDLE
Address	Meaning	Description	
SA45 (bit 2)	Homing direction for synchronous tapping	00: Shorter route 01: Forward rotation 10: Reverse rotation	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
SA45 (bit 3)	Homing direction for synchronous tapping		
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
SA45 (bit 4)	Defines the specified direction as the Z-phase detection direction.		
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	
SA45 (bit 5)	Spindle index gear correction	0: Invalid 1: Valid	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	0, 1	

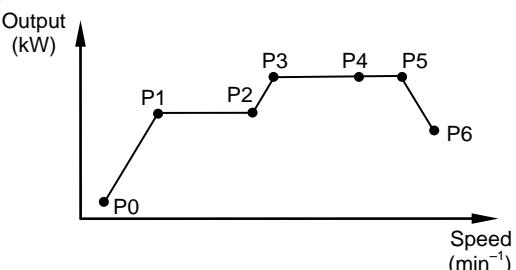
Classification	MACHINE	Display title	SPINDLE
Address	Meaning		Description
SA46	—		<p>Direction of orientation 00: Shorter route 01: Forward rotation 10: Reverse rotation</p> <p>C-axis position control changeover type 00: After return to zero point 01: After deceleration stop</p> <p>Synchronous tapping position control changeover type 0: After return to zero point 1: After deceleration stop</p> <p>Z-phase detection direction 0: Forward rotation 1: Reverse rotation</p> <p>C-axis homing direction 00: Shorter route 01: Forward rotation 10: Reverse rotation</p> <p>Synchronous tapping command polarity 0: Reverse rotation 1: Forward rotation</p>
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	Bit	
	Setting range	Binary, eight digits	
SA47	—		<p>0: Ignoring the spindle/motor gear ratio 1: Considering the spindle/motor gear ratio</p>
	Program type	M, E	
	Conditions	At power on	
	Unit	Bit	
	Setting range	Binary, eight digits	
SA48	Encoder signal input destination		<p>Specify the encoder signal input destination.</p> <p>0: Via the HDLC-connected axis (Spindle AMP feedback data) 1: Direct connection to encoder 1 (ENC1) 2: Direct connection to encoder 2 (ENC2)</p>
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

Classification	MACHINE	Display title	SPINDLE
Address	Meaning	Description	
SA49	Speed attainment detection width	Set the speed attainment detection width. If a value other than 1-99 (%) is set, the value will be regarded as 15 (%).	
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 99	
SA50	Spindle type	Specify the type of spindle corresponding to the displayed SA parameters, by the appropriate numeric value. 0: Axis not specified 1: No. 1 milling spindle 2: No. 1 turning spindle 3: No. 2 milling spindle 4: No. 2 turning spindle 5: No. 3 milling spindle 6: No. 3 turning spindle 7: No. 4 milling spindle 8: No. 4 turning spindle	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 8	
SA51	Number of gears on spindle	Set the number of gears on the spindle. 1. For displaying a gear selection menu in manual operation mode 0, 5 to 8: Without gear 1: 2 gear positions (without neutral) 2: 2 gear positions (with neutral) 3: 3 gear positions (with neutral) 4: 4 gear positions (with neutral) 2. For automatic gear selection with the MAZATROL program (only for milling spindle)	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	0 to 8	
SA52	Turning spindle type	0: Type not set. 1: C-axis type 2: Orientation type	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

Classification	MACHINE	Display title	SPINDLE
Address	Meaning		Description
SA53	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — L coils		<p>Set the 1/4h (1/2h) rated torque for the L coils of the spindle motor.</p> <p>Note :</p> <ol style="list-style-type: none"> Only the 1/2h rated torque, not the 1/4h rated torque, may be known for the particular motor. If that is the case, set the 1/2h rated torque. If coil selection is not possible for the spindle, enter the same value in both SA53 and SA54. An alarm will result if 0 is entered even in either one of the two parameters. If bit 1 in SA45 is set to 1, set the 1/4h rated torque applied when the spindle motor gear position is 1.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 N·m	
	Setting range	0 to 65535	
	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — H coils		<p>Set the 1/4h (1/2h) rated torque for the H coils of the spindle motor.</p> <p>Note :</p> <ol style="list-style-type: none"> Only the 1/2h rated torque, not the 1/4h rated torque, may be known for the particular motor. If that is the case, set the 1/2h rated torque. If coil selection is not possible for the spindle, enter the same value in both SA53 and SA54. An alarm will result if 0 is entered even in either one of the two parameters. If bit 1 in SA45 is set to 1, set the 1/4h rated torque applied when the spindle motor gear position is 2.
SA54	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — Spindle gear position 3		<p>Set the 1/4h rated torque applied when the spindle motor gear position is 3.</p> <p>Note :</p> <ol style="list-style-type: none"> Only the 1/2h rated torque, not the 1/4h rated torque, may be known for the particular motor. If that is the case, set the 1/2h rated torque. This parameter is valid only when the setting of bit 1 in SA45 is 1.
	Program type	M	
	Conditions	Immediate	
	Unit	0.1 N·m	
	Setting range	0 to 65535	
	Spindle 1/4h (1/2h) rated torque for auto-pecking of the cutting load detection type — Spindle gear position 4		<p>Set the 1/4h rated torque applied when the spindle motor gear position is 4.</p> <p>Note :</p> <ol style="list-style-type: none"> Only the 1/2h rated torque, not the 1/4h rated torque, may be known for the particular motor. If that is the case, set the 1/2h rated torque. This parameter is valid only when the setting of bit 1 in SA45 is 1.
SA56	Program type	M	
	Conditions	Immediate	
	Unit	0.1 N·m	
	Setting range	0 to 65535	

Classification	MACHINE	Display title	SPINDLE
Address	Meaning	Description	
SA57	Spindle viscous friction coefficient "cms" for auto-pecking of the cutting load detection type — Range 1 Program type M Conditions Immediate Unit — Setting range ± 99999999	If the "fms" value depends on spindle speeds, specify the gradient. Example : If "fms" is 90000 for an "S" value of 5000 and 140000 for S10000: 	
SA58	Spindle viscous friction coefficient "cms" for auto-pecking of the cutting load detection type — Range 2 Program type M Conditions Immediate Unit — Setting range ± 99999999	Since "cms" = $(140000 - 90000) / (10000 - 5000) = 10$, set "10" in the above example. Note : Set the spindle viscous friction coefficient "cms" when SA45 bit 0 = 1. (Enter, moreover, 1000 times the respective calculation results in this case.)	
SA59	Spindle viscous friction coefficient "cms" for auto-pecking of the cutting load detection type — Range 3 Program type M Conditions Immediate Unit — Setting range ± 99999999		
SA60	Spindle viscous friction coefficient "cms" for auto-pecking of the cutting load detection type — Range 4 Program type M Conditions Immediate Unit — Setting range ± 99999999		

Classification	MACHINE	Display title	SPINDLE
Address	Meaning		Description
SA61	Spindle coulombic friction coefficient "fms" for auto-pecking of the cutting load detection type — Range 1		<p>Set the value where the width of the flat section in the current feedback data matches estimated data.</p> <p>Example : If "fms" is 90000 for an "S" value of 5000 and 140000 for S10000, set "c" in L105.</p>
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	±99999999	
SA62	Spindle coulombic friction coefficient "fms" for auto-pecking of the cutting load detection type — Range 2		<p>Calculate "c" from the linear equation "y = (b/a) x + c". Since "c" = 90000 – (10 × 5000) = 40000, set "40000" in the above example.</p> <p>Note : Set the spindle coulombic friction coefficient "fms" when SA45 bit 0 = 1.</p>
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	±99999999	
SA63	Spindle coulombic friction coefficient "fms" for auto-pecking of the cutting load detection type — Range 3		
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	±99999999	
SA64	Spindle coulombic friction coefficient "fms" for auto-pecking of the cutting load detection type — Range 4		
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	±99999999	

Classification	MACHINE	Display title	SPINDLE																								
Address	Meaning	Description																									
SA65	Cutting force calculation filter for auto-pecking of the cutting load detection type	Set the filter for the data which has been sampled at 3.5-ms intervals. If the entered value is "0", the data actually used will be 4×3.5 (ms).																									
	Program type	M																									
	Conditions	Immediate																									
	Unit	3.5 ms																									
	Setting range	0 to 65535																									
SA66	Maximum permissible speed of milling spindle for polygonal/hobbing machining	If the rotational speed of the milling spindle during polygonal machining exceeds the setting of this parameter, that rotational speed will be clamped at this setting. Also, if the rotational speed of the milling spindle during hobbing exceeds the setting of this parameter, that rotational speed will be clamped at this setting. Note : This parameter is valid only for the milling spindle.																									
	Program type	E																									
	Conditions	At power on																									
	Unit	min^{-1}																									
	Setting range	0 to 65535																									
SA67 to SA73	Rotational speed in the following spindle output diagrams: <ul style="list-style-type: none">• MACHINING NAVIGATION-RESULT• MACHINING NAVIGATION-PPEDITION• Monitoring Functions• POSITION	 <table border="1"> <thead> <tr> <th>Point</th> <th>Cross axis: speed (Unit: min^{-1})</th> <th>Vertical axis: output (Unit: 1/100 kW)</th> </tr> </thead> <tbody> <tr> <td>P0</td> <td>SA67</td> <td>SA74</td> </tr> <tr> <td>P1</td> <td>SA68</td> <td>SA75</td> </tr> <tr> <td>P2</td> <td>SA69</td> <td>SA76</td> </tr> <tr> <td>P3</td> <td>SA70</td> <td>SA77</td> </tr> <tr> <td>P4</td> <td>SA71</td> <td>SA78</td> </tr> <tr> <td>P5</td> <td>SA72</td> <td>SA79</td> </tr> <tr> <td>P6</td> <td>SA73</td> <td>SA80</td> </tr> </tbody> </table>	Point	Cross axis: speed (Unit: min^{-1})	Vertical axis: output (Unit: 1/100 kW)	P0	SA67	SA74	P1	SA68	SA75	P2	SA69	SA76	P3	SA70	SA77	P4	SA71	SA78	P5	SA72	SA79	P6	SA73	SA80	
Point	Cross axis: speed (Unit: min^{-1})	Vertical axis: output (Unit: 1/100 kW)																									
P0	SA67	SA74																									
P1	SA68	SA75																									
P2	SA69	SA76																									
P3	SA70	SA77																									
P4	SA71	SA78																									
P5	SA72	SA79																									
P6	SA73	SA80																									
SA74 to SA80	Output in the following spindle output diagrams: <ul style="list-style-type: none">• MACHINING NAVIGATION-RESULT• MACHINING NAVIGATION-PPEDITION• Monitoring Functions• POSITION	<p>Note :</p> <p>Whether to specify these values for an output diagram at short-time rating or at continuous rating depends on the specifications of the model.</p>																									
	Program type	M, E																									
	Conditions	Immediate																									
	Unit	0.01 kW																									
	Setting range	0 to 99999999																									

Classification	MACHINE	Display title	SPINDLE
Address	Meaning		Description
SA81 to SA83		Set the spindle limit speed at which the time constant for the spindle position control is to be changed	
Spindle limit speed selection for spindle position control time constants SA81: Limit speed 1 SA82: Limit speed 2 SA83: Limit speed 3			
Program type M, E Conditions At power on Unit min⁻¹ Setting range 0 to 99999			
SA84 to SA86		Spindle position control time constants SA84: Time constant 1 SA85: Time constant 2 SA86: Time constant 3	
Program type M, E Conditions At power on Unit ms Setting range 0 to 9999			
SA87 SA88		Spindle speed operating time constant changeover revolutions SA87: Revolutions 1 SA88: Revolutions 2	
Program type M, E Conditions Immediate Unit min⁻¹ Setting range 0 to 99999999			
SA89 to SA91		Spindle speed operating time constant SA89: Time constant 1 SA90: Time constant 2 SA91: Time constant 3	
Program type M, E Conditions Immediate Unit ms Setting range 0 to 99999999		<p>Note : For gears 2 to 8, SA87 or SA88 is internally converted to calculate the changeover revolutions.</p>	

Classification	MACHINE	Display title	SPINDLE
Address	Meaning	Description	
SA92	Z-phase detection speed Program type M, E Conditions Immediate Unit min ⁻¹ Setting range 0 to 99999	Set the maximum revolutions for conducting Z-phase detection.	
SA93	Amount of synchronous tapping zero point shifting Program type M, E Conditions After stop of movement Unit ° Setting range 0 to 359	Set the amount of shifting from the Z-phase when the axis is to be returned to its zero point for synchronous tapping. Shifting is conducted in the Z-phase detection direction (specified in bit 4 of SA46).	
SA94	Homing speed for synchronous tapping Program type M, E Conditions Immediate Unit min ⁻¹ Setting range 0 to 99999	Set the homing speed effective when "After return to zero point" (SA46 bit 2 = 0) is selected as the position-changing type for synchronous tapping.	
SA95	Maximum revolutions in manual operation mode Program type M, E Conditions Immediate Unit min ⁻¹ Setting range 0 to 99999	Set the manual operation mode maximum revolutions.	

Classification	MACHINE	Display title	SPINDLE
Address	Meaning	Description	
SA96	Amount of orientation position shifting	Set the amount of shifting from the zero point of orientation control.	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.00005°	
	Setting range	±7200000	
SA97	Reduction ratio of the synchronous tapping time constant for high-speed synchronous tapping	The synchronous tapping time constant for high-speed synchronous tapping is reduced with respect to the synchronous tapping time constant set in parameter SA33 to SA40 . The value set in SA97 becomes the reduction ratio. Without the high-speed synchronous tapping option or for the axis combination that does not allow high-speed synchronous tapping, normal synchronous tapping is applied, in which case the data setting of SA33 to SA40 is used as it is. High-speed synchronous tapping time constant $= \text{SA33} \times (100 - \text{SA97})/100$	
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
SA99	Orientation time constant	Set the time constant for orientation control. If 0 is set, 300 will be regarded as having been specified.	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	ms	
	Setting range	0 to 30000	

Classification	MACHINE	Display title	SPINDLE																																
Address	Meaning	Description																																	
SA100 to SA106	<p>Rotational speed in the following spindle output diagrams at continuous rating:</p> <ul style="list-style-type: none"> • MACHINING NAVIGATION-RESULT • MACHINING NAVIGATION-PREDICTION • Monitoring Functions • POSITION <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>min^{-1}</td></tr> <tr><td>Setting range</td><td>0 to 65535</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	min^{-1}	Setting range	0 to 65535	<p>Output (kW)</p> <p>Speed (min^{-1})</p> <table border="1"> <tr><td>Point</td><td>Cross axis: speed (Unit: min^{-1})</td><td>Vertical axis: output (Unit: 1/100 kW)</td></tr> <tr><td>P0</td><td>SA100</td><td>SA107</td></tr> <tr><td>P1</td><td>SA101</td><td>SA108</td></tr> <tr><td>P2</td><td>SA102</td><td>SA109</td></tr> <tr><td>P3</td><td>SA103</td><td>SA110</td></tr> <tr><td>P4</td><td>SA104</td><td>SA111</td></tr> <tr><td>P5</td><td>SA105</td><td>SA112</td></tr> <tr><td>P6</td><td>SA106</td><td>SA113</td></tr> </table>	Point	Cross axis: speed (Unit: min^{-1})	Vertical axis: output (Unit: 1/100 kW)	P0	SA100	SA107	P1	SA101	SA108	P2	SA102	SA109	P3	SA103	SA110	P4	SA104	SA111	P5	SA105	SA112	P6	SA106	SA113	
Program type	M, E																																		
Conditions	Immediate																																		
Unit	min^{-1}																																		
Setting range	0 to 65535																																		
Point	Cross axis: speed (Unit: min^{-1})	Vertical axis: output (Unit: 1/100 kW)																																	
P0	SA100	SA107																																	
P1	SA101	SA108																																	
P2	SA102	SA109																																	
P3	SA103	SA110																																	
P4	SA104	SA111																																	
P5	SA105	SA112																																	
P6	SA106	SA113																																	
SA107 to SA113	<p>Output in the following spindle output diagrams at continuous rating:</p> <ul style="list-style-type: none"> • MACHINING NAVIGATION-RESULT • MACHINING NAVIGATION-PREDICTION • Monitoring Functions • POSITION <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.01 kW</td></tr> <tr><td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.01 kW	Setting range	0 to 99999999	<p>Note :</p> <ol style="list-style-type: none"> 1. Set values for spindle output diagrams at continuous rating. 2. This parameter is valid for models that allow you to switch between load meter ratings on the POSITION display. 																									
Program type	M, E																																		
Conditions	Immediate																																		
Unit	0.01 kW																																		
Setting range	0 to 99999999																																		
SA114	<p>Spindle speed supervisory mode 2 for safety supervision</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>min^{-1}</td></tr> <tr><td>Setting range</td><td>0 to 10000</td></tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	min^{-1}	Setting range	0 to 10000	<p>Set the spindle speed to be monitored in safety supervisory mode 2.</p> <p>If the operating speed of the spindle exceeds the set value in safety supervisory mode 2, this will cause a safety supervisory alarm and bring the machine to an emergency stop.</p>																									
Program type	M, E																																		
Conditions	After stop of movement																																		
Unit	min^{-1}																																		
Setting range	0 to 10000																																		
SA115	<p>Spindle speed supervisory mode 3 for safety supervision</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>min^{-1}</td></tr> <tr><td>Setting range</td><td>0 to 10000</td></tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	min^{-1}	Setting range	0 to 10000	<p>Set the spindle speed to be monitored in safety supervisory mode 3.</p> <p>If the operating speed of the spindle exceeds the set value in safety supervisory mode 3, this will cause a safety supervisory alarm and bring the machine to an emergency stop.</p> <p>Assign a value greater than the speed value to be monitored in mode 2 (i.e., specified in SA114).</p>																									
Program type	M, E																																		
Conditions	After stop of movement																																		
Unit	min^{-1}																																		
Setting range	0 to 10000																																		

Classification	MACHINE	Display title	SPINDLE
Address	Meaning	Description	
SA116	Spindle safety clamping mode 2 for safety supervision	<p>Set the spindle safety clamping speed to be monitored in safety supervisory mode 2.</p> <p>In safety supervisory mode 2 and while a safety clamping request is in effect, the spindle is decelerated to the set speed.</p> <p>Assign a value smaller than the speed value to be monitored in mode 2 (i.e., specified in SA114).</p>	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	min ⁻¹	
	Setting range	0 to 10000	
SA117	Spindle safety clamping mode 3 for safety supervision	<p>Set the spindle safety clamping speed to be monitored in safety supervisory mode 3.</p> <p>In safety supervisory mode 3 and while a safety clamping request is in effect, the spindle is decelerated to the set speed.</p> <p>Assign a value smaller than the speed value to be monitored in mode 3 (i.e., specified in SA115).</p> <p>Assign a value greater than the speed value to be monitored in mode 2 (i.e., specified in SA116).</p>	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	min ⁻¹	
	Setting range	0 to 10000	
SA118	Selecting the spindle door of the spindle whose speed is to be monitored	<p>Select the door group to which the spindle belongs in safety supervisory mode.</p> <p>If SP129: SFNC9/bit F is OFF, speed monitoring will not be conducted, irrespective of the setting of this parameter.</p> <p>[Examples of setting]</p> <p>0000: The spindle does not belong to any door.</p> <p>0001: The spindle belongs to the door-1 group.</p> <p>0002: The spindle belongs to the door-2 group.</p> <p>0003: The spindle belongs to the door-1/2 group.</p>	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	—	
SA119	Deceleration judgment coefficient on safety speed clamping of the spindle to be supervised for safety	<p>For safety speed clamping in safety supervisory mode, when a clamping request is input, a clamping signal will be output after the spindle has decelerated to a safety clamping speed. This parameter specifies to what additional percentage of the safety clamping speed the spindle is to be decelerated as the output timing of the speed clamping signal.</p> <p>If 0 is set, the speed clamping signal will be output when the spindle decelerates to a 10% additional speed (i.e., 110% of the safety clamping speed).</p>	
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	%	
	Setting range	0 to 100	

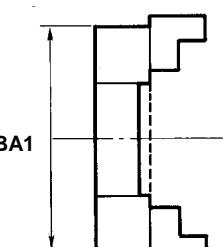
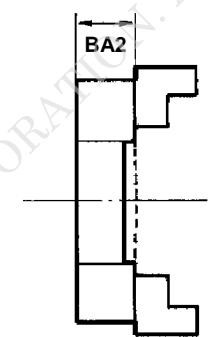
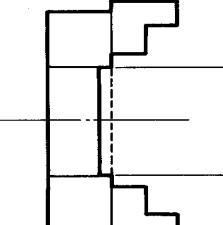
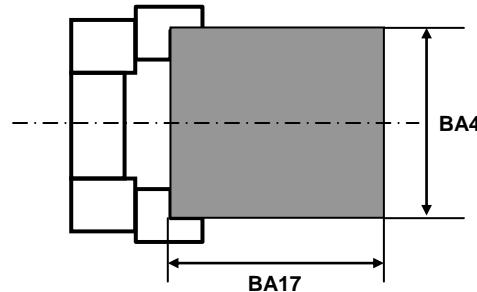
Classification	MACHINE	Display title	SPINDLE
Address	Meaning		Description
SA120	PLG pulse rate for spindle index gear tooth correction		Set the number of PLG teeth per revolution. If a value less than zero is set, 1024 will be regarded as having been set.
	Program type	M, E	
	Conditions	Immediate	
	Unit	Teeth	
	Setting range	-99999999 to 99999999	
SA121 to SA128	Amount of branching point correction for spindle index gear tooth SA121: Branching point [1] SA122: Branching point [2] SA123: Branching point [3] SA124: Branching point [4] SA125: Branching point [5] SA126: Branching point [6] SA127: Branching point [7] SA128: Branching point [8]		Set the amount of correction at the branching point where one PLG gear tooth is split into eight equal segments.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.0001°	
	Setting range	-99999999 to 99999999	
SA129 SA131	Notch filter frequency for cutting-load detection pecking cycle SA129: #1 notch filter frequency SA131: #2 notch filter frequency		Set the notch filter frequency for the cutting-load detection pecking cycle.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.1 Hz	
	Setting range	0 to 65535	
SA130 SA132	Notch filter depth for cutting-load detection pecking cycle SA130: #1 notch filter depth SA132: #2 notch filter depth		Set the notch filter depth to be used in the cutting-load detection pecking cycle.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 7	

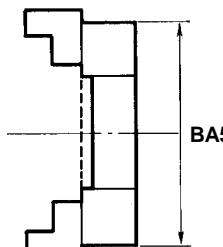
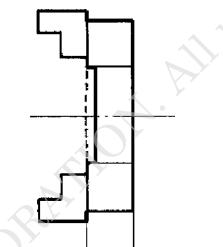
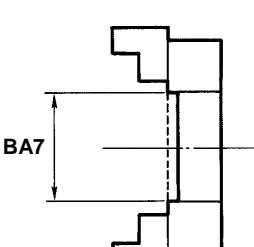
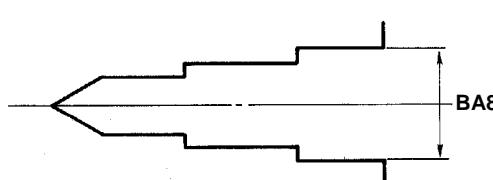
Classification	MACHINE	Display title	SPINDLE
Address	Meaning	Description	
SA133	Breakage detection torque for cutting-load detection pecking cycle		Set the detection torque to be used when the breakage detection distance (l) is specified for the cutting-load detection pecking cycle.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.01 N·m	
	Setting range	0 to 65535	
SA134	Maximum spindle orientation speed		Set the maximum spindle orientation speed. Note : 1. Some other functions (such as safety speed clamping control based on a PLC ladder) may cause the spindle orientation speed to be clamped at a speed lower than the setting of this parameter. 2. Immediately after power-on, the Z-phase is detected, so the spindle is oriented at the maximum speed specified in parameter SA92 , not SA134 . 3. Clamping based on this parameter will be invalid if the setting is 0.
	Program type	M, E	
	Conditions	Immediate	
	Unit	min ⁻¹	
	Setting range	0 to 99999999	
SA136	Selection of a door for signal input on the drive side (spindle) in safety supervisory mode		Set which door signal is input for the axis with door signal input on the drive side (spindle) in safety supervisory mode. 1: Door 1 signal 2: Door 2 signal 4: Door 3 signal 8: Door 4 signal
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 255	
SA138 SA139	Number of gears SA138: Number of spindle gears SA139: Number of motor gears		Set the number of gears. 0 : No gears 1 or more : Actual number of gears
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 99999999	

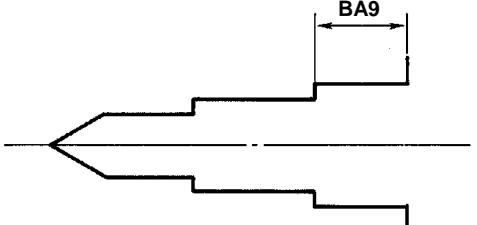
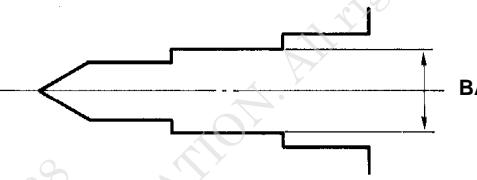
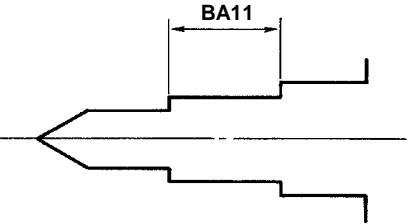
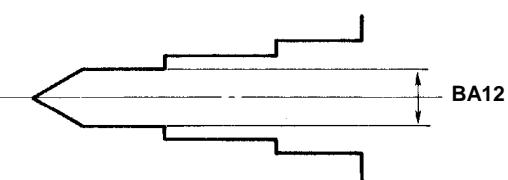
Classification	MACHINE	Display title	SPINDLE
Address	Meaning		Description
SA140	Turret indexing gear ratio		Set the gear ratio for turret indexing.
	Program type	M, E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	—	
SA143 (bit 1)	Waiting for phase compensation completion at the start of high-speed synchronous tapping		Set whether or not to wait spindle orientation completion at the start of high-speed synchronous tapping. 0: Do not wait 1: Wait
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
SA143 (bit 3)	Spindle speed for the orient command given during spindle rotation		When the orient command is given during spindle rotation 0: Hold the spindle speed at the time of the orient command. 1: Accelerate the spindle speed up to the maximum speed for orientation.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
SA144 (bit 4)	Encoder polarity for the spindle rotation for normal thread cutting		0: Spindle cycle counter increase 1: Spindle cycle counter decrease
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	

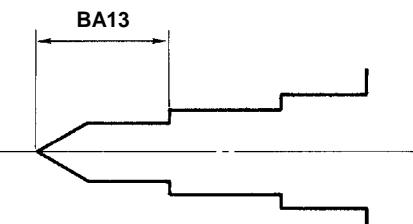
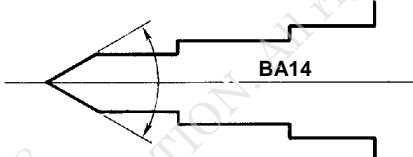
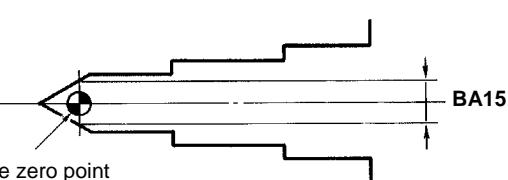
Classification	MACHINE	Display title	SPINDLE	
Address	Meaning	Description		
SA144 (bit 5)	Gear conversion during spindle rotation valid		Gear conversion during spindle rotation valid	
	Program type	M, E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
SA144 (bit 6)	Spindle gear changeover valid		Spindle gear changeover valid	
	Program type	M, E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		
SA144 (bit 7)	Turret indexing valid		Turret indexing valid	
	Program type	M, E		
	Conditions	Immediate		
	Unit	Bit		
	Setting range	0, 1		

2-3-16 Machine parameter BARRIER (BA)

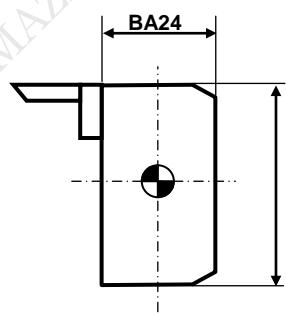
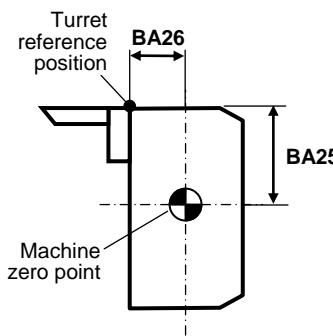
Classification	MACHINE	Display title	BARRIER
Address	Meaning		Description
BA1	Chuck outside diameter (for chuck barrier) — No. 1 turning spindle		Setting of chuck outside diameter of the No. 1 turning spindle  NM211-00312
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	
BA2	Chuck width (for chuck barrier) — No. 1 turning spindle		Setting of chuck width of the No. 1 turning spindle  NM211-00313
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	
BA3	Chuck inside diameter (for chuck barrier) — No. 1 turning spindle		Setting of chuck inside diameter of the No. 1 turning spindle  NM211-00314
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	
BA4	EIA program workpiece outside diameter		Specify the workpiece outside diameter to be used for the EIA program. 
	Program type	E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA5	Chuck outside diameter (for chuck barrier) — No. 2 turning spindle	Setting of chuck outside diameter of the No. 2 turning spindle  Note : For single turning-spindle specifications, this parameter is invalid.	NM211-00312
BA6	Chuck width (for chuck barrier) — No. 2 turning spindle	Setting of chuck width of the No. 2 turning spindle  Note : For single turning-spindle specifications, this parameter is invalid.	NM211-00313
BA7	Chuck inside diameter (for chuck barrier) — No. 2 turning spindle	Setting of chuck inside diameter of the No. 2 turning spindle  Note : For single turning-spindle specifications, this parameter is invalid.	NM211-00314
BA8	Tail body outside diameter (for tail barrier)	Setting of tail body outside diameter  Note : For single turning-spindle specifications, this parameter is invalid.	NM211-00315

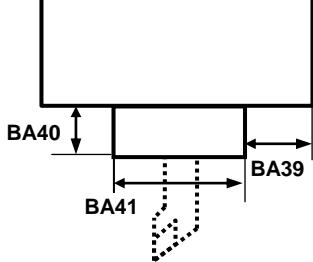
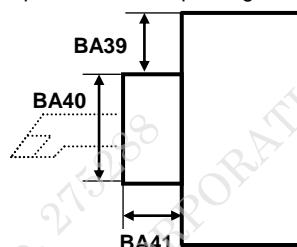
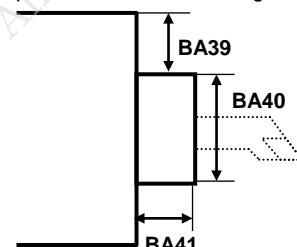
Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA9	Tail body length (for tail barrier)	Setting of tail body length 	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	NM211-00316
BA10	Tail spindle outside diameter (for tail barrier)	Setting of tail spindle outside diameter 	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	NM211-00317
BA11	Length with tail spindle at back end (for tail barrier)	Setting of length with tail spindle at back end 	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	NM211-00318
BA12	Tail head outside diameter (for tail barrier)	Setting of tail head outside diameter 	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	NM211-00319

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA13	Tail head length (for tail barrier)	Setting of tail head length 	
	Program type	M, E	NM211-00320
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	
BA14	Tail head taper angle (for tail barrier)	Setting of tail head taper angle 	
	Program type	M, E	NM211-00321
	Conditions	Immediate	
	Unit	0.001°	
	Setting range	0 to 180000	
BA15	Tail head biting diameter (for tail barrier)	Setting of biting diameter when tail head is used 	
	Program type	M, E	NM211-00322
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA16 to BA18	EIA tail barrier Tail dimensions	<p>Set the dimensional data for forming tail barriers using an EIA/ISO program.</p> <p><During tail operation></p> <p><During tail reversing></p> <p>BA16: Tail extruding length BA17: Workpiece length (Note 2) BA18: Distance from the machine zero point to the leading edge position during tail reversing</p> <p>Note :</p> <ol style="list-style-type: none"> 1. ±999999999 for BA18. 2. BA17 is valid for MAZATROL programs as well, when ONLY MILL is specified for the common unit of the MAZATROL program. 	
Program type	E		
Conditions	Immediate		
Unit	0.001 mm/0.0001 in		
Setting range	0 to 99999999 (Note 1)		
BA19	Distance from the Z-axis machine zero point to the spindle edge — No. 1 turning spindle	<p>Specify the distance from the machine zero point of the Z-axis to the edge of the No. 1 turning spindle.</p> <p>Note :</p> <p>Enter the distance with minus sign for the machine zero point in the minus direction with respect to the spindle edge.</p>	
Program type	M, E		
Conditions	Immediate		
Unit	0.001 mm/0.0001 in		
Setting range	±99999999		
BA20	Distance from the Z-axis machine zero point to the spindle edge — No. 2 turning spindle	<p>Specify the distance from the machine zero point of the Z-axis to the edge of the No. 2 turning spindle.</p> <p>Note :</p> <p>Enter the distance with minus sign for the machine zero point in the minus direction with respect to the spindle edge.</p>	
Program type	M, E		
Conditions	Immediate		
Unit	0.001 mm/0.0001 in		
Setting range	±99999999		

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA21	Jaw number for EIA program barrier — No. 1 turning spindle		Specify the jaw number that has been registered on the CHUCK JAW DATA display to be referred to in the EIA program when forming a jaw barrier for the No. 1 turning spindle.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	1 to 44	
BA22	Jaw number for EIA program barrier — No. 2 turning spindle		Specify the jaw number that has been registered on the CHUCK JAW DATA display to be referred to in the EIA program when forming a jaw barrier for the No. 2 turning spindle.
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	1 to 44	
BA23 BA24	Turret dimensions		Set the turret dimensions for the turret-type tool post. 
	Program type	E	BA23: Outside diameter of the turret BA24: Width of the turret
BA25 BA26	Turret reference position		Specify the turret reference position from the machine zero point for the turret-type tool post. 
	Program type	E	BA25: Turret reference position X BA26: Turret reference position Z
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA27 BA30 BA33 BA36	Tool holder mounting position BA27 - Type 1 BA30 - Type 2 BA33 - Type 3 BA36 - Type 4 Program type M Conditions Immediate Unit 0.001 mm/0.0001 in Setting range ±99999999	Setting of tool holder mounting position. When plus data is used, the tool holder is mounted horizontally, and minus data downward. Example : Type 1 Diagram showing a tool holder mounted horizontally at a height of BA27 (> 0) above the Turret reference position. A dashed line indicates the horizontal center of the tool holder. Diagram showing a tool holder mounted vertically downwards at a height of BA27 (< 0) below the Turret reference position. A dashed line indicates the vertical center of the tool holder. Same for types 2, 3, 4	Turret reference positon BA27 (> 0) BA27 (< 0) NM211-00327'
BA28 BA31 BA34 BA37	Tool holder width in X-axis direction BA28 - Type 1 BA31 - Type 2 BA34 - Type 3 BA37 - Type 4 Program type M Conditions Immediate Unit 0.001 mm/0.0001 in Setting range 0 to 99999999	Setting tool holder width in X-axis direction Example : Type 1 Diagram showing a tool holder with a width of BA28 (where BA27 > 0) centered horizontally relative to the Turret reference position. A dashed line indicates the horizontal center of the tool holder. Diagram showing a tool holder with a width of BA28 (where BA27 < 0) centered vertically downwards relative to the Turret reference position. A dashed line indicates the vertical center of the tool holder. Same for types 2, 3, 4	Turret reference position BA28 (where BA27 > 0) BA28 (where BA27 < 0) NM211-00328'
BA29 BA32 BA35 BA38	Tool holder width in Z-axis direction BA29 - Type 1 BA32 - Type 2 BA35 - Type 3 BA38 - Type 4 Program type M Conditions Immediate Unit 0.001 mm/0.0001 in Setting range 0 to 99999999	Setting tool holder width in Z-axis direction Example : Type 1 Diagram showing a tool holder with a width of BA29 (where BA27 > 0) centered horizontally relative to the Turret reference position. A dashed line indicates the horizontal center of the tool holder. Diagram showing a tool holder with a width of BA29 (where BA27 < 0) centered vertically downwards relative to the Turret reference position. A dashed line indicates the vertical center of the tool holder. Same for types 2, 3, 4	Turret reference position BA29 (where BA27 > 0) BA29 (where BA27 < 0) NM211-00329'

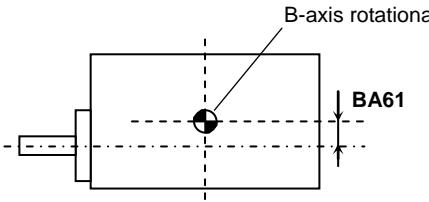
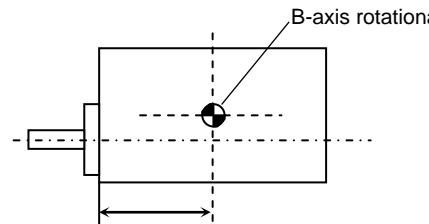
Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA39 to BA41	EIA tool barrier Holder dimensions	<p>Set the holder shape data for forming tool barriers using an EIA/ISO program.</p> <p><Holder-under type> Input BA39 with a minus sign.</p> 	
		<p><Holder-side type (0° type)> Input BA39 with a plus sign. Input BA41 with a plus sign.</p> 	
		<p><Holder-side type (180° type)> Input BA39 with a plus sign. Input BA41 with a minus sign.</p> 	
		<p>BA39: EIA tool barrier, Holder mounting position BA40: EIA tool barrier, X-axis width of the holder BA41: EIA tool barrier, Z-axis width of the holder</p>	
		<p>Note : 0 to 99999999 for BA40.</p>	
BA42	Barrier type	<p>Select the type of barrier to be made valid.</p> <p>0: Type with head B-axis (tool rotational B-axis) [INTEGREX type] 1: Type without head B-axis (tool rotational B-axis)</p>	
		<p>Program type M, E</p>	
		<p>Conditions Immediate</p>	
		<p>Unit —</p>	
		<p>Setting range 0 to 99999999</p>	

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA43	First tool number (in the 1st set of tools)	Set the first tool number assigned to the first set of tools.	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	1 to 4000	
BA44	Number of tools (in the 1st set of tools)	Set the number of tools assigned as the first set of tools.	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	1 to 960	
BA45	First tool number (in the 2nd set of tools)	Set the first tool number assigned to the second set of tools.	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	1 to 4000	
BA46	Number of tools (in the 2nd set of tools)	Set the number of tools assigned as the second set of tools.	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	1 to 960	

Classification	MACHINE	Display title	BARRIER
Address	Meaning		Description
BA47	Turret type Program type Conditions Unit Setting range	M, E At power on — 0 to 4	Set the type of turret: 0: ATC 1: Turret 2: Lower turret 3: Opposed turret 4: Steady rest
BA48	Axis name of the head to be rotated Program type Conditions Unit Setting range	M, E After stop of movement — &41 to &5A	Set the axis name of the head axis to be rotated. Set "&42" if the head axis to be rotated is the B-axis.
BA49	Axis number of the inclined axis (Inclined-axis control) Program type Conditions Unit Setting range	M, E At power on — 0 to 16	Set the axis number of the Y-axis to be controlled using inclined-axis control functions. The inclined-axis control is invalid when this parameter is set to 0.
BA50	Fundamental axis number (Inclined-axis control) Program type Conditions Unit Setting range	M, E At power on — 0 to 16	Set the axis number of the X-axis to be controlled using inclined-axis control functions. The inclined-axis control is invalid when this parameter is set to 0.

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA51	Vector of virtual Y (Inclined-axis control)	<p>Specify one of the vectors created by the triangles formed by inclination angles.</p> <p>BA51: Vector of the inclined axis (virtual Y) in the rectangular coordinate system</p> <p>BA52: Vector of the fundamental axis (X-axis) corresponding to the inclined axis</p> <p>BA53: Vector of the real axis corresponding to the inclined axis</p>	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	±99999999	
BA52	Vector of real X (Inclined-axis control)	Set the value of BA51/BA52 assuming that 10000000 is assigned to BA53 .	
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	±99999999	
BA53	Vector of real Y (Inclined-axis control)		
	Program type	M, E	
	Conditions	At power on	
	Unit	—	
	Setting range	±99999999	
BA54	Selection of work spindle for hobbing	<p>Set the axis number of the work spindle to be used for hobbing.</p> <p>The hobbing is invalid when this parameter is set to 0.</p>	
	Program type	E	
	Conditions	After stop of movement	
	Unit	—	
	Setting range	0 to 16	

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA55	Turning spindle number for polygonal machining (D1)	Set the turning spindle to be used in the D1 command mode of polygonal machining. 0: Turning spindle No. 1 1: Turning spindle No. 2 2: Turning spindle No. 3 3: Turning spindle No. 4 -1: Invalid	
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	-1 to 3	
BA56	Turning spindle number for polygonal machining (D2)	Set the turning spindle to be used in the D2 command mode of polygonal machining. 0: Turning spindle No. 1 1: Turning spindle No. 2 2: Turning spindle No. 3 3: Turning spindle No. 4 -1: Invalid	
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	-1 to 3	
BA57	Turning spindle number for polygonal machining (D3)	Set the turning spindle to be used in the D3 command mode of polygonal machining. 0: Turning spindle No. 1 1: Turning spindle No. 2 2: Turning spindle No. 3 3: Turning spindle No. 4 -1: Invalid	
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	-1 to 3	
BA58	Turning spindle number for polygonal machining (D4)	Set the turning spindle to be used in the D4 command mode of polygonal machining. 0: Turning spindle No. 1 1: Turning spindle No. 2 2: Turning spindle No. 3 3: Turning spindle No. 4 -1: Invalid	
	Program type	E	
	Conditions	Immediate	
	Unit	—	
	Setting range	-1 to 3	

Classification	MACHINE	Display title	BARRIER											
Address	Meaning	Description												
BA59	Spindle forward rotation M-code for tapping cycle Program type M, E Conditions Immediate Unit — Setting range 0 to 255	<ul style="list-style-type: none"> If the command block of G284/G288 (machining G-codes, F30 = 0) or of G84/G88 (turning G-codes, F30 = 1) does not contain spindle forward/reverse rotation M-codes, one of the following M-codes will be output, depending on the status of bit 0 in parameter SU153: <table border="1"> <tr> <td></td><td></td><td>BA59 = 3 BA60 = 4</td><td>BA59 = 203 BA60 = 204</td></tr> <tr> <td rowspan="2">SU153 Bit 0</td><td>0</td><td>Tapping cycle of turning</td><td>Tapping cycle of milling</td></tr> <tr> <td>1</td><td>Inverse tapping cycle of turning</td><td>Inverse tapping cycle of milling</td></tr> </table> <ul style="list-style-type: none"> If 0 is set in BA59, 3 will be regarded as having been set, and M03 will be output. If 0 is set in BA60, 4 will be regarded as having been set, and M04 will be output. 			BA59 = 3 BA60 = 4	BA59 = 203 BA60 = 204	SU153 Bit 0	0	Tapping cycle of turning	Tapping cycle of milling	1	Inverse tapping cycle of turning	Inverse tapping cycle of milling	
		BA59 = 3 BA60 = 4	BA59 = 203 BA60 = 204											
SU153 Bit 0	0	Tapping cycle of turning	Tapping cycle of milling											
	1	Inverse tapping cycle of turning	Inverse tapping cycle of milling											
BA60	Spindle reverse rotation M-code for tapping cycle Program type — Conditions — Unit — Setting range —													
BA61	Amount of runout of the B-axis center Program type M, E Conditions Immediate Unit 0.001 mm Setting range -1000 to 1000	<p>Enter the distance from the rotational center of the B-axis to the tool center.</p>  <p>Note : Enter the distance with a minus sign for B-axis rotational center above the tool center position.</p>												
BA62	Amount of offset for the B-axis — spindle distance Program type M, E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range ±99999999	<p>Enter the distance from the rotational center of the B-axis to the spindle edge.</p> 												

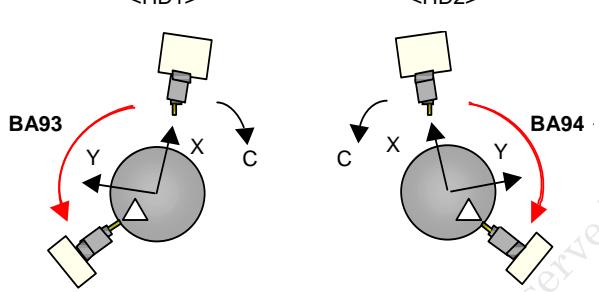
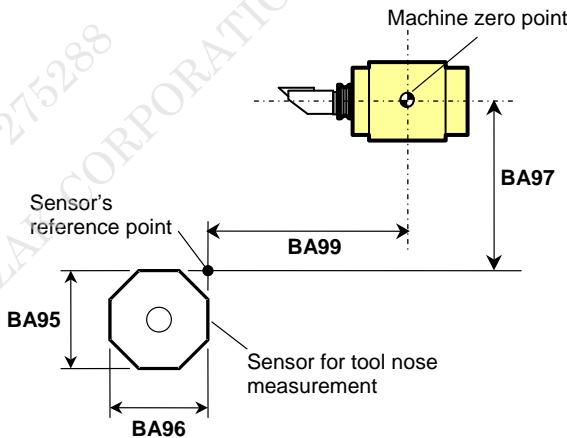
Classification	MACHINE	Display title	BARRIER
Address	Meaning		Description
BA63	Holder angle of angle tool holder		<p>Set the holder angle of the angle tool holder. According to the value set in this parameter, the holder angle is set in HLD.TYPE on the TOOL DATA display.</p> <p>BA63 = 45</p>
	Program type	M	
	Conditions	Immediate	
	Unit	°	
	Setting range	0 to 90	
BA64	B-axis tool reference position X		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	±99999999	
BA65	B-axis tool reference position Z		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	±99999999	
BA66 BA67	G37 deceleration area G37 measuring area		<p>Set the deceleration area and measuring area in the G37 command.</p> <p>BA66: Set the distance between the starting point of movement at the measuring rate, and the measuring point. This value will be used when argument R is omitted from the G37 command.</p> <p>BA67: Set the moving distance in measuring feed mode. This value will be used when argument D is omitted from the G37 command.</p>
	Program type	E	
	Conditions	After stop of movement	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	

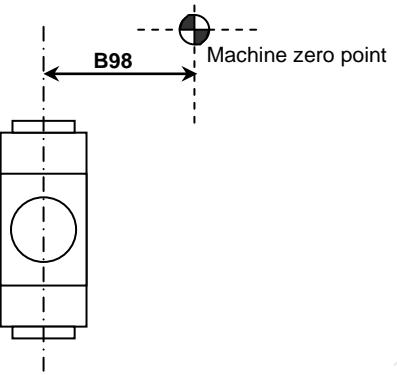
Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA68 BA69	G36 deceleration area G36 measuring area Program type Conditions Unit Setting range	<p>Set the deceleration area and measuring area in the G36 command.</p> <p>BA68: Set the distance between the starting point of movement at the measuring rate, and the measuring point. This value will be used when argument R is omitted from the G36 command.</p> <p>BA69: Set the moving distance in measuring feed mode. This value will be used when argument D is omitted from the G36 command.</p>	
BA70	Distance between the reference points on both turrets (radius value) Program type Conditions Unit Setting range	<ul style="list-style-type: none"> Workpiece zero point: Workpiece edge center Tool position reference point: Reference point on the turret Turret clearance: Distance between the reference points on both turrets (radius value) Workpiece offset: Workpiece zero point – Reference point of reference turret tool position Tool position: Tool position reference point – Tool nose position <p><Schematic diagram></p>	
BA71	System number to be used when argument L is omitted from G112 Program type Conditions Unit Setting range	0: System 1 1: System 2 2: System 3 3: System 4	

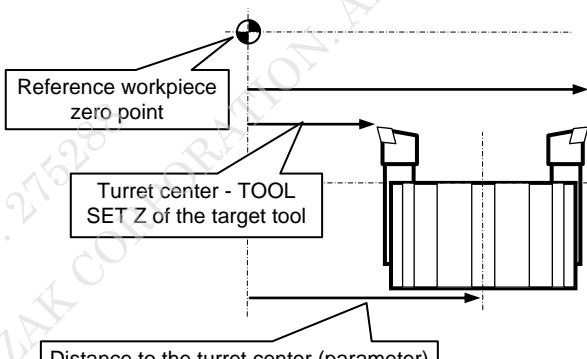
Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA72	Feed override for retracting the tap	<p>Set the overriding value for retracting the tap.</p> <p>Tap retracting feed = [Programmed feed for the interrupted synchronous tapping] × BA72/100</p> <p>BA72 is processed as 100 (%) when the setting is equal to zero (0) or greater than 100.</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	%	
	Setting range	0 to 100	
BA73	Barrier valid/invalid 1 (chuck, sub-chuck, tailstock)	<p>0: Chuck, sub-chuck and tailstock invalid</p> <p>1: Chuck valid</p> <p>2: Sub-chuck valid</p> <p>3: Tailstock valid</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 3	
BA74	Barrier valid/invalid 2 (Lower turret, steady rest)	<p>0: Lower turret and steady rest invalid</p> <p>1: Lower turret valid</p> <p>2: Steady rest valid</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	

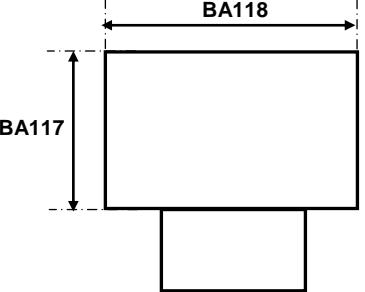
Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA75 to BA78	Barrier setup turret reference position	<p>Set the reference position when it is viewed from the machine zero point.</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	± 99999999	
BA79 to BA82	Barrier setup chuck reference position 1	<p>Set the reference position 1 of the chucks with respect to the machine zero point of the upper turret. Use J109 to J112 for machines with the No. 2 spindle set as System 1.</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	± 99999999	
BA83 to BA86	Barrier setup chuck reference position 2	<p>Set the chuck reference position 2 to be used to set barrier around jaws for a program without jaw No. setting. Use J113 to J116 for machines with the No. 2 spindle set as System 1.</p>	
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	± 99999999	

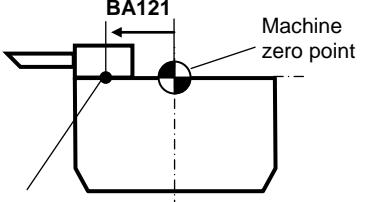
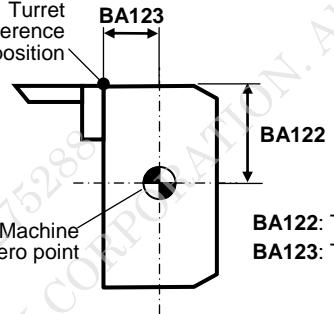
Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA87 to BA90	Barrier reference position for a powered tailstock Program type M, E Conditions Immediate Unit 0.001 mm/0.0001 in Setting range ± 99999999	<p>Set the reference position for barrier area of a powered tailstock.</p> <p>Use coordinates with respect to the machine zero point of the upper turret to set the position of the tailstock being not used for machining.</p> <p>When the tailstock is used, the barrier will be provided at the position shifted through the distance of R10366 (lower-order)/R10367 (higher-order) in the Z-axis direction, from the position specified above.</p> <p>For machines with multiple systems use only parameters for System 1.</p>	
BA91	Distance from spindle edge to partition plate Program type M, E Conditions Immediate Unit 0.001 mm/0.0001 in Setting range ± 99999999	<p>Set the reference position of the partition plate.</p>	
BA92	Central position X when viewed from machine zero point (Barrier function) Program type M, E Conditions Immediate Unit 0.001 mm/0.0001 in Setting range —	<p>Set the central position X when viewed from the machine zero point.</p>	

Classification	MACHINE	Display title	BARRIER		
Address	Meaning	Description			
BA93 BA94	BA93: Upper/lower turret tool angle difference (HD1) BA94: Upper/lower turret tool angle difference (HD2)		When using the C-axes of the HD1/HD2 in the lower turret system, set the offset angle with respect to the upper turret, for the lower turret system. 		
	Program type	M, E			
	Conditions	Immediate			
	Unit	0.0001°			
	Setting range	±99999999			
BA95 BA96 BA97 BA99 BA100 BA102 BA103 BA104	BA95: Sensor width along the X-axis BA96: Sensor width along the Z-axis BA97: X-coordinate of the sensor's reference point BA99: Z-coordinate of the sensor's reference point BA103: Sensor width along the X-axis (for lower turret/HD2) BA104: Sensor width along the Z-axis (for lower turret/HD2) BA100: X-coordinate of the sensor's reference point (for lower turret/HD2) BA102: Z-coordinate of the sensor's reference point (for lower turret/HD2)	Use BA95 and BA96 to set the dimensions of the sensor for tool nose measurement. BA97 and BA99 are provided to set in machine coordinates the reference point of the sensor (see the diagram below). Example : 1. For horizontal type 			
	Program type	M			
	Conditions	Immediate			
	Unit	0.0001 mm/0.00001 in			
	Setting range	±99999999			

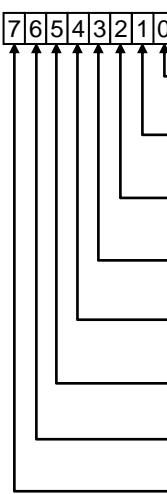
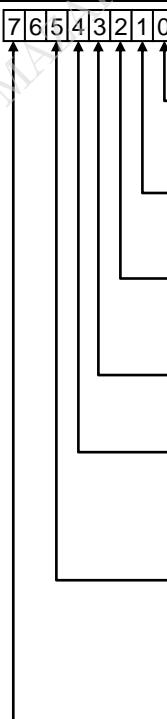
Classification	MACHINE	Display title	BARRIER							
Address	Meaning	Description								
BA98 BA101	<p>BA98: Y-coordinate of the sensor's reference point BA101: Y-coordinate of the sensor's reference point (for lower turret/HD2)</p> <table border="1"> <tr><td>Program type</td><td>M</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.0001 mm/0.00001 in</td></tr> <tr><td>Setting range</td><td>± 99999999</td></tr> </table>	Program type	M	Conditions	Immediate	Unit	0.0001 mm/0.00001 in	Setting range	± 99999999	<p>Setting of sensor reference point Y coordinate</p> 
Program type	M									
Conditions	Immediate									
Unit	0.0001 mm/0.00001 in									
Setting range	± 99999999									
BA105	<p>Adjustment "FdT" for servo axis thrust hold</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>0.0005 mm/0.00005 in</td></tr> <tr><td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	0.0005 mm/0.00005 in	Setting range	0 to 99999999	<p>Set adjustment "FdT" for holding the thrust of the servo axis. Set a value less than BA108 × BA106/100. If any other value is set, the thrust may be unstable.</p>
Program type	M, E									
Conditions	After stop of movement									
Unit	0.0005 mm/0.00005 in									
Setting range	0 to 99999999									
BA106	<p>Dead zone</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>%</td></tr> <tr><td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	%	Setting range	0 to 99999999	<p>Set the dead zone for the drooping level of the servo axis thrust. A drooping level change less than BA108 × BA106/100 is ignored.</p>
Program type	M, E									
Conditions	After stop of movement									
Unit	%									
Setting range	0 to 99999999									
BA107	<p>Filter</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>After stop of movement</td></tr> <tr><td>Unit</td><td>1/3.5 ms</td></tr> <tr><td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	M, E	Conditions	After stop of movement	Unit	1/3.5 ms	Setting range	0 to 99999999	<p>Set the monitoring time for changes in the drooping level of the servo axis thrust.</p>
Program type	M, E									
Conditions	After stop of movement									
Unit	1/3.5 ms									
Setting range	0 to 99999999									

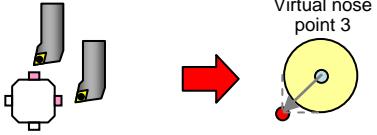
Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA108	Servo axis thrust hold droop Program type M, E Conditions After stop of movement Unit 0.00005 mm Setting range 0 to 99999999	Set the drooping level for holding the servo axis thrust. This drooping level is held during the application of the servo axis thrust.	
BA109	Offset amount (X) during automatic tool setting value calculation Program type M, E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range ±99999999	Specify the axis offset amount applied during automatic calculation of the tool setting value for the backface tool, which is difficult to measure using the TOOL EYE. This automatic calculation is done based on the tool setting value for the target tool measured in the opposite position.	
BA110	Offset amount (Z) during automatic tool setting value calculation Program type M, E Conditions Immediate Unit 0.0001 mm/0.00001 in Setting range ±99999999	$\text{TOOL SET X} = \text{Tool setting value X for target tool} + \text{Parameter for offset amount X}$ $\text{TOOL SET Z} = (\text{Parameter for distance to center of turret} \times (-1) \times 2) - \text{Tool setting value Z for target tool} + \text{Parameter for offset amount Z}$	
BA111 to BA116	Pre-interpolation acceleration/deceleration time constant for time constant changeover M-code command BA111: For M881 command BA112: For M882 command BA113: For M883 command BA114: For M884 command BA115: For M885 command BA116: For M886 command Program type M, E Conditions After stop of movement Unit mm/min Setting range 0 to 999999	Set the pre-interpolation acceleration/deceleration time constant for a time constant changeover M-code command. If zero is assigned to this parameter, the pre-interpolation acceleration/deceleration time constant will not be changed, even when a time constant changeover M-code command is assigned. The pre-interpolation acceleration/deceleration time constant existing before the time constant changeover M-code command is assigned will be held.	

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA117	Outside diameter of the turret for Virtual Machining		Set the outside diameter and width for the turret which is to be indicated on the VIRTUAL MACHINING display. 
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	
BA118	Width of the turret for Virtual Machining		
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	
BA119	Selecting a head having a lower-turret TOOL EYE		0: Head 1 1: Head 2 2: Head 1/2
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0 to 2	
BA120	Mounting position X for an assembly tool model		Set the X-axis position of mounting the HOLDER part of an assembly tool model.
	Program type	M, E	
	Conditions	Immediate	
	Unit	0.001 mm/0.0001 in	
	Setting range	0 to 99999999	

Classification	MACHINE	Display title	BARRIER								
Address	Meaning	Description									
BA121	<p>Mounting position Z for an assembly tool model</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 99999999	<p>Set the Z-axis position of mounting the HOLDER part of an assembly tool model.</p>  <p>Assembly tool model mounting position</p>	
Program type	M, E										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 99999999										
BA122 BA123	<p>For VIRTUAL MACHINING or 3D MONITOR display Turret reference position</p> <table border="1"> <tr><td>Program type</td><td>M, E</td></tr> <tr><td>Conditions</td><td>Immediate</td></tr> <tr><td>Unit</td><td>0.001 mm/0.0001 in</td></tr> <tr><td>Setting range</td><td>0 to 99999999</td></tr> </table>	Program type	M, E	Conditions	Immediate	Unit	0.001 mm/0.0001 in	Setting range	0 to 99999999	<p>Specify the turret reference position from the machine zero point for the turret-type tool post (for VIRTUAL MACHINING or 3D MONITOR display).</p>  <p>BA122: Turret reference position X BA123: Turret reference position Z</p>	
Program type	M, E										
Conditions	Immediate										
Unit	0.001 mm/0.0001 in										
Setting range	0 to 99999999										
BA124	—	Fixed value (0)									
	<table border="1"> <tr><td>Program type</td><td>—</td></tr> <tr><td>Conditions</td><td>—</td></tr> <tr><td>Unit</td><td>—</td></tr> <tr><td>Setting range</td><td>—</td></tr> </table>	Program type	—	Conditions	—	Unit	—	Setting range	—		
Program type	—										
Conditions	—										
Unit	—										
Setting range	—										

Classification	MACHINE	Display title	BARRIER								
Address	Meaning	Description									
BA125	EIA barrier activation	<p>Set the types of tailstock, tool holder and chuck for which the EIA barriers are to be activated.</p> <table border="0"> <tr> <td>7</td> <td>6</td> <td>5</td> <td>4</td> <td>3</td> <td>2</td> <td>1</td> <td>0</td> </tr> </table> <ul style="list-style-type: none"> { Tailstock usage valid/invalid (Note 1) <ul style="list-style-type: none"> 0: Invalid 1: Valid { Tailstock present/absent (Note 2) <ul style="list-style-type: none"> 0: Absent 1: Present { Tool holder present/absent <ul style="list-style-type: none"> 0: Absent 1: Present { Barrier tool nose position <ul style="list-style-type: none"> 0: Tool length correction 1: Current tool nose Setting of BA11 is handled as tailstock extruding length { Execution of spindle revolution M-code specified in same block as synchronous tapping <ul style="list-style-type: none"> 0: Invalid 1: Valid { Barrier cancellation check (see Note 3 and 4) <ul style="list-style-type: none"> 0: Valid 1: Invalid { Missing jaw number check (see Note 3) <ul style="list-style-type: none"> 0: Valid 1: Invalid <p>Note :</p> <ol style="list-style-type: none"> 1. "Tailstock usage invalid" does not mean canceling the tail barriers; it means that the tail will move away prior to machining. 2. Bit 1 in this parameter is valid for the MAZATROL program as well. If bit 1 is set to 0, the tailstock function itself will become invalid. For a machine having a tailstock, be sure to set up 1 at bit 1. 3. Valid only for MAZATROL programs. 4. Bit 6 of this parameter is valid only when the setup data has been entered. 	7	6	5	4	3	2	1	0	
7	6	5	4	3	2	1	0				
	Program type	E									
	Conditions	Immediate									
	Unit	Bit									
	Setting range	Binary, eight digits									

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA126	—	 <p>System to be made valid without system selection for queuing</p> <p>Y-axis interference type 0: Type A 1: Type B</p> <p>Y-axis moving range display 0: Invalid 1: Valid</p> <p>Tool command scheme (Note) 0: M06 required. 1: M06 not required.</p> <p>FLASH tool valid/invalid 0: Invalid 1: Valid</p> <p>Milling spindle orientation command (M219) 0: Output 1: No output</p> <p>System without turret</p> <p>Shape of the tailstock center for Virtual Machining 0: Single stage 1: Twin stage</p>	
	Program type	E	Note : T-command operation is defined as follows by the setting of bit 3: <ul style="list-style-type: none">If the setting of bit 3 is 0<ul style="list-style-type: none">TOT◆M6: Replaces the current tool with ◆ and indexes ◆ as the next tool.TOM6: Replaces the current tool with O.TO: Indexes O as the next tool.If the setting of bit 3 is 1<ul style="list-style-type: none">TO□: Replaces the current tool with O and set □ as the tool offset number.
	Conditions	Immediate	
	Unit	Bit	
	Setting range	Binary, eight digits	
BA127		 <p>M249 command for preparing for tool change before moving the tool to the tool change position using a MAZATROL program 0: No output 1: Output</p> <p>For programs of Z-offset scheme: Fixed value (1) For other programs: Invalid</p> <p>Simultaneous output of TTM6M200 and TTM6M202 0: No simultaneous output 1: Simultaneous output</p> <p>NC tailstock (servo axis thrust control) 0: Invalid 1: Valid</p> <p>M90/M91 output for machines with turret (for QTN II series) 0: Output 1: No output</p> <p>Type of the clamping and unclamping M-codes that the NC automatically output for machines equipped with a spindle head of swiveling type. 0: M-codes for machine tools of composite machining 1: M-codes for machining centers</p> <p>Whether or not to read the current settings (tailstock position) on the SET UP MANAG. display at the beginning of the execution of each program 0: No 1: Yes</p>	
	Program type	M (Bit 3, 7: M, E)	
	Conditions	Immediate (Bit 3: After stop of movement)	
	Unit	Bit	
	Setting range	Binary, eight digits	

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA128 (bit 0)	NC tailstock pushing direction NC tailstock pushing direction	NC tailstock pushing direction 0: Negative direction 1: Positive direction	
	Program type M Conditions Immediate Unit Bit Setting range 0, 1		
BA128 (bit 1)	Method of automatic determination of the virtual nose point for tool measuring unit	<p>Select the method of automatic determination of the virtual nose point for a MAZATROL program's tool measuring unit (TOOL MES) to be executed with the TOOL EYE.</p> <p>0: With reference to the PART and CUT DIR. items on the TOOL DATA display</p> <p>Example : PART: OUT, CUT DIR.: LEFT ↘</p>  <p>1: With reference to PTN (measuring pattern: from [TOOL EYE #1] to [TOOL EYE #4]) in the sequence data of the tool measuring unit</p> <p>Example : PTN: [TOOL EYE #1]</p> 	
BA128 (bit 2)	Display of THRUST F. and HORSE PW for the AFC function in the first set of tool data (BA43, BA44) Display of THRUST F. and HORSE PW for the AFC function in the first set of tool data (BA43, BA44)	<p>Display of THRUST F. and HORSE PW for the AFC function in the first set of tool data (BA43, BA44)</p> <p>0: Valid 1: Invalid</p>	
	Program type M Conditions Immediate Unit Bit Setting range 0, 1		

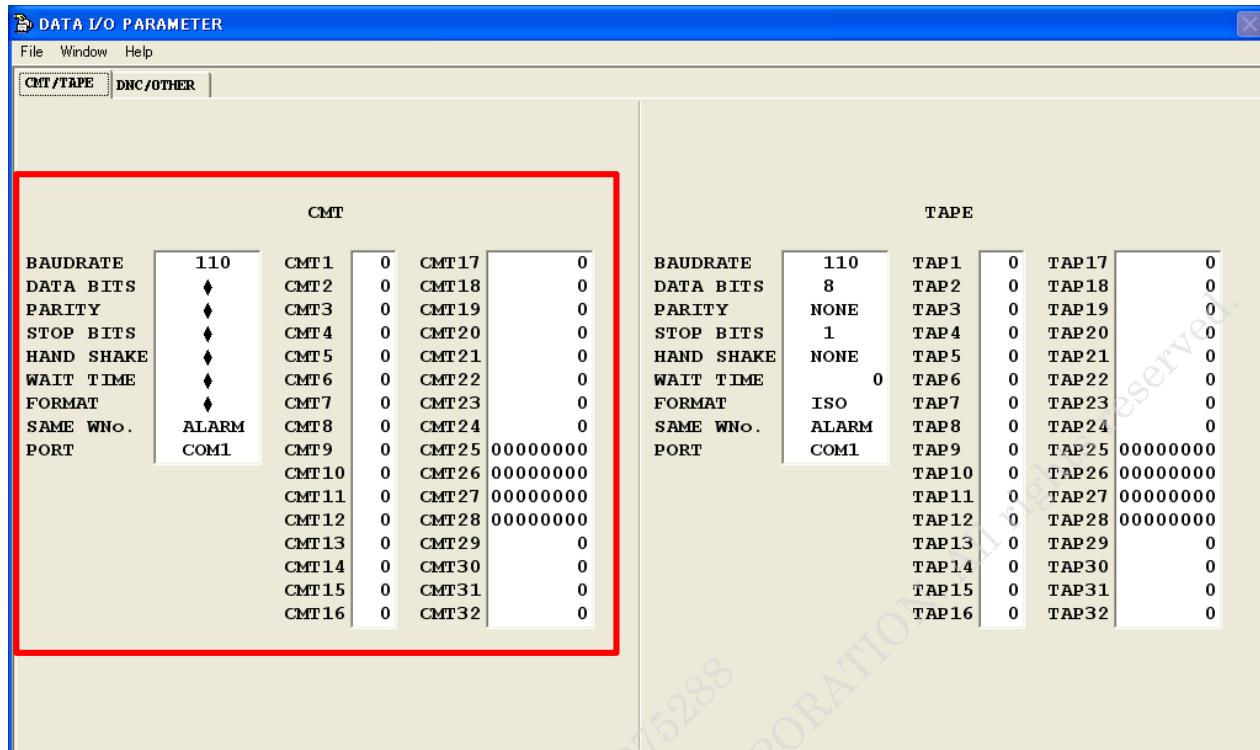
Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA128 (bit 3)	Display of THRUST F. and HORSE PW for the AFC function in the second set of tool data (BA45, BA46) Program type M Conditions Immediate Unit Bit Setting range 0, 1	Display of THRUST F. and HORSE PW for the AFC function in the second set of tool data (BA45, BA46) 0: Valid 1: Invalid	
BA128 (bit 6)	Turret lathe with an ATC unit Program type E Conditions Immediate Unit Bit Setting range 0, 1	0: Invalid 1: Valid	
BA128 (bit 7)	Display/No display of [TURRET UNCLAMP] for machines with an ATC-type turret Program type E Conditions At pwer on Unit Bit Setting range 0, 1	Display of [TURRET UNCLAMP] for machines with an ATC-type turret 0: Invalid 1: Valid Note : Setting of bit 7 is valid only for machines without lower turret.	
BA129 (bit 1)	— Program type — Conditions — Unit — Setting range —	Fixed value	

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA129 (bit 2)	Fixed G0 inclination control in the G68/G68.5 mode		Select the inclination to be used for fixed G0 inclination control (option) in the G68/G68.5 mode when parameter M1 (rapid feed rate) and parameter N1 (rapid-feed time constant) are used. 0: Uses the minimum inclination among the actual axes after G68/G68.5 conversion. 1: Uses the minimum inclination among the virtual axes before G68/G68.5 conversion. (*) * If the move command specifies one axis, the inclination of the specified axis is used.
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
BA129 (bit 3)	Selection of whether or not to measure the milling tool length with the tool measuring unit		Select whether or not to measure the milling tool length with the tool measuring unit. 0: When a milling tool is selected, its length is not measured and the alarm 714 ILLEGAL SEQUENCE DATA is raised. 1: The tool length is measured. When BA129 bit 3 is 0, no measurement is performed on models in which parameters are set as described below. <ul style="list-style-type: none">• BA47 = 0 (turret type: ATC)• BA48 = &42 (axis of the head to be rotated: B)• K52 = 0 Note : For INTEGREX i with a workpiece hand, even if BA129 bit 3 is 0, the length of a milling tool can be measured.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
BA129 (bit 4)	Display of [BARRIER CANCEL] menu item with [BARRIER CANCEL TEACH] made valid		Display of [BARRIER CANCEL] menu item with [BARRIER CANCEL TEACH] made valid 0: Invalid 1: Valid
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	
BA129 (bit 5)	Handling of measurement speed of 1 in/min or higher during laser tool length/diameter measurement with inch-specification machines		0: Drop the fractional portion. 1: Do not drop the fractional portion. Examples of measurement speed during the execution of this measurement are given below.
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
		BA129 (bit 5)	
		Set speed	0 1
		Example (1) 0.9 in/min	0.9 in/min 0.9 in/min
		Example (2) 1.1 in/min	1.0 in/min 1.1 in/min

Classification	MACHINE	Display title	BARRIER
Address	Meaning	Description	
BA131 (bit 0)	Multi-magazine number display	Multi-magazine number indication on the POSITION display 0: Not set 1: Set	
	Program type	M, E	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1	

2-3-17 Data I/O parameter CMT parameter (CMT)

Parameter setting



D740H0001E

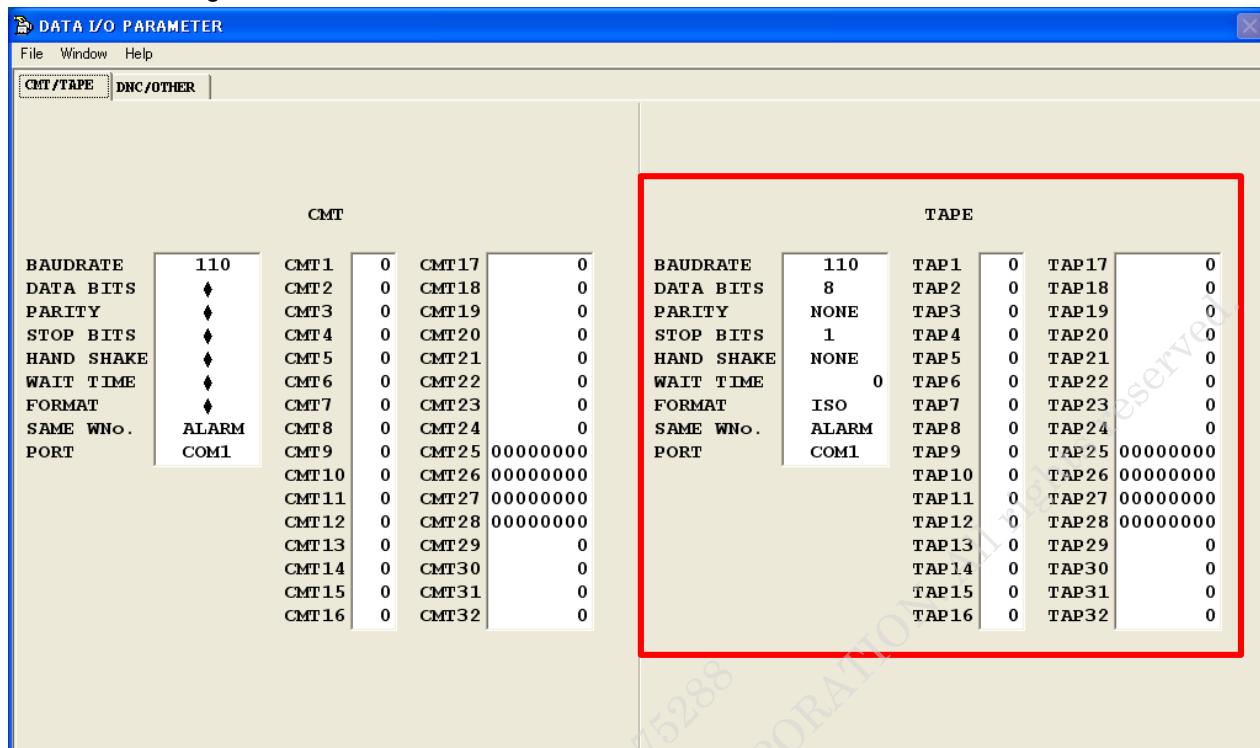
Classification	DATA I/O	Display title	CMT								
Meaning			Description								
BAUDRATE			Baud rate for RS-232C interface								
Set values			<table border="1"> <tr><td>110</td><td>4800</td></tr> <tr><td>300</td><td>9600</td></tr> <tr><td>1200</td><td>19200</td></tr> <tr><td>2400</td><td></td></tr> </table>	110	4800	300	9600	1200	19200	2400	
110	4800										
300	9600										
1200	19200										
2400											
Program type			M, E								
Conditions			At I/O startup								
Unit			—								
Setting range			110 to 19200								
SAME WNo.			Type of processing to be executed if the machining program of an existing work number is to be loaded								
Set values			<table border="1"> <tr> <td>ALARM</td> <td>Issues an alarm if the work number already exists.</td> </tr> <tr> <td>LOAD</td> <td>Overrides the program if the work number already exists.</td> </tr> </table>	ALARM	Issues an alarm if the work number already exists.	LOAD	Overrides the program if the work number already exists.				
ALARM	Issues an alarm if the work number already exists.										
LOAD	Overrides the program if the work number already exists.										
Program type			M, E								
Conditions			At I/O startup								
Unit			—								
Setting range			—								

Classification	DATA I/O	Display title	CMT										
Meaning		Description											
PORT		CMT port selection											
		<table border="1"> <thead> <tr> <th>Set values</th><th>Description</th></tr> </thead> <tbody> <tr> <td>COM1</td><td>CF22 serial ch3</td></tr> <tr> <td>COM2</td><td>CF22 serial ch4</td></tr> <tr> <td>COM3</td><td>USB-RS232C conversion connector</td></tr> <tr> <td>COM4</td><td>USB-RS232C conversion connector</td></tr> </tbody> </table>		Set values	Description	COM1	CF22 serial ch3	COM2	CF22 serial ch4	COM3	USB-RS232C conversion connector	COM4	USB-RS232C conversion connector
Set values	Description												
COM1	CF22 serial ch3												
COM2	CF22 serial ch4												
COM3	USB-RS232C conversion connector												
COM4	USB-RS232C conversion connector												
Program type	M, E												
Conditions	At I/O startup												
Unit	—												
Setting range	—												

Classification	DATA I/O	Display title	CMT
Address	Meaning	Description	
CMT1 to CMT32	—	Invalid	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

2-3-18 Data I/O parameter TAPE parameter (TAP)

Parameter setting



D740H0002E

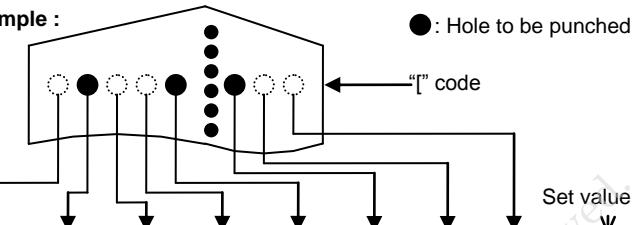
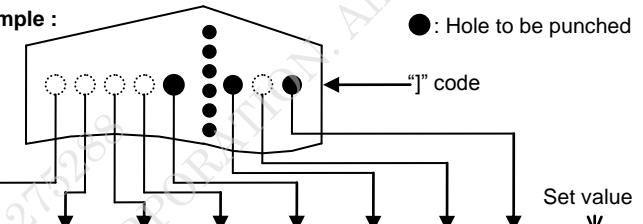
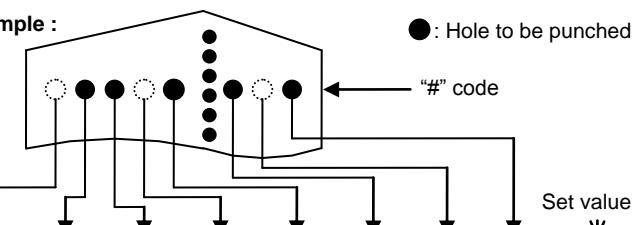
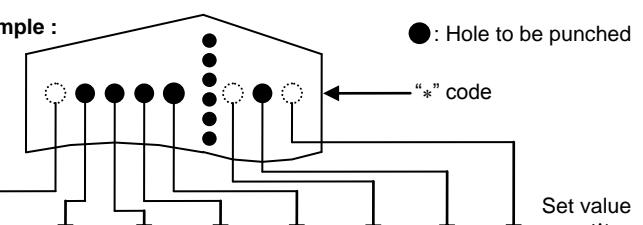
Classification	DATA I/O	Display title	TAPE								
Meaning		Description									
BAUDRATE		Baud rate for RS-232C interface									
Program type	M, E	Set values	<table border="1"> <tr><td>110</td><td>4800</td></tr> <tr><td>300</td><td>9600</td></tr> <tr><td>1200</td><td>19200</td></tr> <tr><td>2400</td><td></td></tr> </table>	110	4800	300	9600	1200	19200	2400	
110	4800										
300	9600										
1200	19200										
2400											
Conditions	At I/O startup										
Unit	—										
Setting range	110 to 19200										
DATA BITS		Number of data bits (parameter for RS-232C interface initialization)									
Program type	M, E	Set values	<table border="1"> <tr><td>7</td></tr> <tr><td>8</td></tr> </table>	7	8						
7											
8											
Conditions	At I/O startup										
Unit	—										
Setting range	7, 8										

Classification	DATA I/O	Display title	TAPE								
Meaning			Description								
PARITY			Parity check (parameter for RS-232C interface initialization)								
			Set values								
			<table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>NONE</td></tr> <tr><td>ODD</td></tr> <tr><td>EVEN</td></tr> </table>	NONE	ODD	EVEN					
NONE											
ODD											
EVEN											
Program type	M, E										
Conditions	At I/O startup										
Unit	—										
Setting range	—										
STOP BITS			Number of stop bits (parameter for RS-232C interface initialization)								
			Set values								
			<table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>1</td></tr> <tr><td>1.5</td></tr> <tr><td>2</td></tr> </table>	1	1.5	2					
1											
1.5											
2											
Program type	M, E										
Conditions	At I/O startup										
Unit	—										
Setting range	—										
HAND SHAKE			This parameter is used to select the method of handshaking to control the state of data transfer between the NC system and connected device.								
			<table border="1" style="margin-left: auto; margin-right: auto;"> <tr><th>Set values</th><th>Description</th></tr> <tr><td>NONE</td><td>No control</td></tr> <tr><td>DC CONTROL</td><td>Complies with control code DC1 through DC4</td></tr> <tr><td>RTS/CTS</td><td>Complies with device connection RTS/CTS.</td></tr> </table>	Set values	Description	NONE	No control	DC CONTROL	Complies with control code DC1 through DC4	RTS/CTS	Complies with device connection RTS/CTS.
Set values	Description										
NONE	No control										
DC CONTROL	Complies with control code DC1 through DC4										
RTS/CTS	Complies with device connection RTS/CTS.										
Program type	M, E										
Conditions	At I/O startup										
Unit	—										
Setting range	—										
WAIT TIME			The waiting time for replies from the connected device during inputting or outputting. An alarm occurs if this time elapses following the final reply.								
Program type	M, E										
Conditions	At I/O startup										
Unit	0.1 s										
Setting range	0 to 65535										

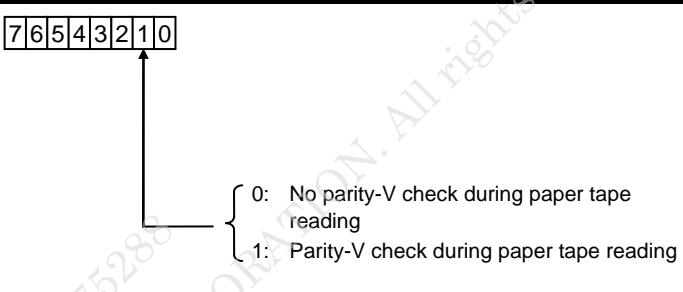
Classification	DATA I/O	Display title	TAPE																						
		Meaning																							
FORMAT			Selection of paper tape puncher output code																						
			<table border="1"> <thead> <tr> <th>Set values</th><th>Description</th></tr> </thead> <tbody> <tr> <td>ISO</td><td>Paper tape punching in ISO code</td></tr> <tr> <td>EIA</td><td>Paper tape punching in EIA code</td></tr> <tr> <td>ASCII</td><td>Paper tape punching in ASCII code</td></tr> </tbody> </table>	Set values	Description	ISO	Paper tape punching in ISO code	EIA	Paper tape punching in EIA code	ASCII	Paper tape punching in ASCII code														
Set values	Description																								
ISO	Paper tape punching in ISO code																								
EIA	Paper tape punching in EIA code																								
ASCII	Paper tape punching in ASCII code																								
Program type			M, E																						
Conditions			At I/O startup																						
Unit			—																						
Setting range			—																						
SAME WNo.			Type of processing to be executed if the machining program of an existing work number is to be loaded																						
			<table border="1"> <thead> <tr> <th>Set values</th><th>Description</th></tr> </thead> <tbody> <tr> <td>ALARM</td><td>Issues an alarm if the received work number already exists.</td></tr> <tr> <td>LOAD</td><td>Overrides the program if the received work number already exists.</td></tr> </tbody> </table>	Set values	Description	ALARM	Issues an alarm if the received work number already exists.	LOAD	Overrides the program if the received work number already exists.																
Set values	Description																								
ALARM	Issues an alarm if the received work number already exists.																								
LOAD	Overrides the program if the received work number already exists.																								
Program type			M, E																						
Conditions			At I/O startup																						
Unit			—																						
Setting range			—																						
PORT			Tape port selection																						
			<table border="1"> <thead> <tr> <th rowspan="2">Set values</th><th rowspan="2">Data I/O</th><th colspan="2">Tape operation</th></tr> <tr> <th>TAP24 = 0</th><th>TAP24 = 1</th></tr> </thead> <tbody> <tr> <td>COM1</td><td>CF22 serial ch3</td><td>CF22 serial ch3</td><td>CF22 serial ch3</td></tr> <tr> <td>COM2</td><td>CF22 serial ch4</td><td>CF22 serial ch4</td><td>CF22 serial ch4</td></tr> <tr> <td>COM3</td><td>USB-RS232C conversion connector</td><td>Invalid</td><td>USB-RS232C conversion connector</td></tr> <tr> <td>COM4</td><td>USB-RS232C conversion connector</td><td>Invalid</td><td>USB-RS232C conversion connector</td></tr> </tbody> </table>	Set values	Data I/O	Tape operation		TAP24 = 0	TAP24 = 1	COM1	CF22 serial ch3	CF22 serial ch3	CF22 serial ch3	COM2	CF22 serial ch4	CF22 serial ch4	CF22 serial ch4	COM3	USB-RS232C conversion connector	Invalid	USB-RS232C conversion connector	COM4	USB-RS232C conversion connector	Invalid	USB-RS232C conversion connector
Set values	Data I/O	Tape operation																							
		TAP24 = 0	TAP24 = 1																						
COM1	CF22 serial ch3	CF22 serial ch3	CF22 serial ch3																						
COM2	CF22 serial ch4	CF22 serial ch4	CF22 serial ch4																						
COM3	USB-RS232C conversion connector	Invalid	USB-RS232C conversion connector																						
COM4	USB-RS232C conversion connector	Invalid	USB-RS232C conversion connector																						
Program type			M, E																						
Conditions			At I/O startup																						
Unit			—																						
Setting range			—																						

Classification	DATA I/O	Display title	TAPE														
Address	Meaning	Description															
TAP1	Type of terminator	<table border="1"> <thead> <tr> <th>Set values</th><th>Terminator</th></tr> </thead> <tbody> <tr><td>0</td><td>Without terminator</td></tr> <tr><td>1</td><td>EOB or EOR</td></tr> <tr><td>2</td><td>EOB only</td></tr> <tr><td>3</td><td>EOR only</td></tr> <tr><td>4</td><td>One character of your choice</td></tr> <tr><td>5</td><td>Two characters of your choice</td></tr> </tbody> </table>		Set values	Terminator	0	Without terminator	1	EOB or EOR	2	EOB only	3	EOR only	4	One character of your choice	5	Two characters of your choice
Set values	Terminator																
0	Without terminator																
1	EOB or EOR																
2	EOB only																
3	EOR only																
4	One character of your choice																
5	Two characters of your choice																
Program type	M, E																
Conditions	At I/O startup																
Unit	—																
Setting range	0 to 5																
TAP2	Terminator code 1	Effective only when TAP1 is set to 4 or 5.															
	Program type	M, E															
	Conditions	At I/O startup															
	Unit	—															
	Setting range	0 to 255															
TAP3	Terminator code 2	Effective only when TAP1 is set to 5.															
	Program type	M, E															
	Conditions	At I/O startup															
	Unit	—															
	Setting range	0 to 255															
TAP4	Output of CR during ISO code punching	<p>This parameter is used to specify whether or not CR is to be placed in front of LF (separation of blocks) during ISO code punching.</p> <p>0: No placement of CR 1: Placement of CR</p>															
	Program type	M, E															
	Conditions	At I/O startup															
	Unit	—															
	Setting range	0, 1															

Classification	DATA I/O	Display title	TAPE																																	
Address	Meaning	Description																																		
TAP5	DC code parity		This parameter is used to specify whether or not a parity bit is to be assigned to the DC code to be output.																																	
			<table border="1"> <thead> <tr> <th>Set values</th> <th>Parity</th> <th colspan="8">Hole-punching pattern of DC3 code</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>No assignment</td> <td></td><td></td><td></td><td>●</td><td>●</td><td></td><td></td><td></td><td>●</td><td>●</td> </tr> <tr> <td>1</td> <td>Assignment</td> <td>●</td><td></td><td></td><td>●</td><td>●</td><td></td><td></td><td></td><td>●</td><td>●</td> </tr> </tbody> </table>	Set values	Parity	Hole-punching pattern of DC3 code								0	No assignment				●	●				●	●	1	Assignment	●			●	●				●
Set values	Parity	Hole-punching pattern of DC3 code																																		
0	No assignment				●	●				●	●																									
1	Assignment	●			●	●				●	●																									
		<p>Note :</p> <p>This parameter is valid only when HAND SHAKE is set to DC CONTROL.</p>																																		
Program type	M, E																																			
Conditions	At I/O startup																																			
TAP6	Unit	—																																		
	Setting range	0, 1																																		
	Feed section DC code output		Select whether or not DC2 and DC4 codes are to be output to the feed sections which will be generated at the beginning and end of paper tape punching.																																	
			<p>Example :</p> <pre> graph LR A[Feed] --> B[EOR] B --> C["(Significant information)"] C --> D[EOR] D --> E[Feed] style A fill:#fff,stroke:#000,stroke-width:1px style B fill:#fff,stroke:#000,stroke-width:1px style C fill:#fff,stroke:#000,stroke-width:1px style D fill:#fff,stroke:#000,stroke-width:1px style E fill:#fff,stroke:#000,stroke-width:1px style A fill:#fff,stroke:#000,stroke-width:1px style B fill:#fff,stroke:#000,stroke-width:1px style C fill:#fff,stroke:#000,stroke-width:1px style D fill:#fff,stroke:#000,stroke-width:1px style E fill:#fff,stroke:#000,stroke-width:1px </pre>																																	
	Program type	M, E																																		
TAP6	Conditions	At I/O startup																																		
	Unit	—																																		
	Setting range	0 to 3																																		
			<table border="1"> <thead> <tr> <th>Set values</th> <th>Description</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>Neither DC2 nor DC4 is output.</td> </tr> <tr> <td>1</td> <td>Only DC2 is output.</td> </tr> <tr> <td>2</td> <td>Only DC4 is output.</td> </tr> <tr> <td>3</td> <td>Both DC2 and DC4 are output.</td> </tr> </tbody> </table>	Set values	Description	0	Neither DC2 nor DC4 is output.	1	Only DC2 is output.	2	Only DC4 is output.	3	Both DC2 and DC4 are output.																							
Set values	Description																																			
0	Neither DC2 nor DC4 is output.																																			
1	Only DC2 is output.																																			
2	Only DC4 is output.																																			
3	Both DC2 and DC4 are output.																																			
		<p>Note :</p> <p>This parameter is valid only when HAND SHAKE is set to DC CONTROL.</p>																																		
TAP7	DSR output flow control selection		Select whether or not DSR flow control is valid for serial communication during tape operation. 0: Valid 1: Invalid																																	
Note :		This parameter setting is enabled when the cycle start button is pressed to start tape operation.																																		

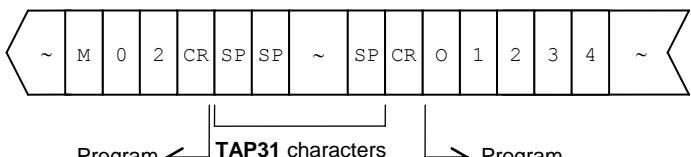
Classification	DATA I/O	Display title	TAPE	
Address		Meaning		
TAP9		<p>"[" code for paper tape reader/puncher for EIA</p> <p>Program type M, E</p> <p>Conditions At I/O startup</p> <p>Unit —</p> <p>Setting range 0 to 255</p>	<p>This parameter is used to set a hole-punching pattern for the character code "[" onto a paper tape reader/puncher using EIA. Set an eight-digit binary number in decimal form.</p> <p>Example :</p>  $(0 \times 2^7) + (1 \times 2^6) + (0 \times 2^5) + (0 \times 2^4) + (1 \times 2^3) + (1 \times 2^2) + (0 \times 2^1) + (0 \times 2^0) = 76$ <p>MPL068</p>	
TAP10		<p">"]" code for paper tape reader/puncher for EIA <p>Program type M, E</p> <p>Conditions At I/O startup</p> <p>Unit —</p> <p>Setting range 0 to 255</p> </p">	<p>This parameter is used to set a hole-punching pattern for the character code "]" onto a paper tape reader/puncher using EIA. Set an eight-digit binary number in decimal form.</p> <p>Example :</p>  $(0 \times 2^7) + (0 \times 2^6) + (0 \times 2^5) + (0 \times 2^4) + (1 \times 2^3) + (1 \times 2^2) + (0 \times 2^1) + (1 \times 2^0) = 13$ <p>MPL069</p>	
TAP11		<p>"#" code for paper tape reader/puncher for EIA</p> <p>Program type M, E</p> <p>Conditions At I/O startup</p> <p>Unit —</p> <p>Setting range 0 to 255</p>	<p>This parameter is used to set a hole-punching pattern for the character code "#" onto a paper tape reader/puncher using EIA. Set an eight-digit binary number in decimal form.</p> <p>Example :</p>  $(0 \times 2^7) + (1 \times 2^6) + (1 \times 2^5) + (0 \times 2^4) + (1 \times 2^3) + (1 \times 2^2) + (0 \times 2^1) + (1 \times 2^0) = 109$ <p>MPL070</p>	
TAP12		<p>"*" code for paper tape reader/puncher for EIA</p> <p>Program type M, E</p> <p>Conditions At I/O startup</p> <p>Unit —</p> <p>Setting range 0 to 255</p>	<p>This parameter is used to set a hole-punching pattern for the character code "*" onto a paper tape reader/puncher using EIA. Set an eight-digit binary number in decimal form.</p> <p>Example :</p>  $(0 \times 2^7) + (1 \times 2^6) + (1 \times 2^5) + (1 \times 2^4) + (1 \times 2^3) + (0 \times 2^2) + (1 \times 2^1) + (0 \times 2^0) = 122$ <p>MPL071</p>	

Classification	DATA I/O	Display title	TAPE
Address	Meaning	Description	
TAP13	"=" code for paper tape reader/puncher for EIA	<p>This parameter is used to set a hole-punching pattern for the character code "=" onto a paper tape reader/puncher using EIA. Set an eight-digit binary number in decimal form.</p> <p>Example :</p> <p>(0 × 2⁷) + (1 × 2⁶) + (0 × 2⁵) + (1 × 2⁴) + (1 × 2³) + (0 × 2²) + (1 × 2¹) + (1 × 2⁰) = 91 MPL072</p>	
TAP14	:	<p>This parameter is used to set a hole-punching pattern for the character code ":" onto a paper tape reader/puncher using EIA. Set an eight-digit binary number in decimal form.</p> <p>Example :</p> <p>(0 × 2⁷) + (1 × 2⁶) + (0 × 2⁵) + (0 × 2⁴) + (0 × 2³) + (1 × 2²) + (1 × 2¹) + (0 × 2⁰) = 70 MPL073</p>	
TAP15	(" code for paper tape reader/puncher for EIA	<p>This parameter is used to set a hole-punching pattern for the character code "(" onto a paper tape reader/puncher using EIA. Set an eight-digit binary number in decimal form.</p> <p>Example :</p> <p>(0 × 2⁷) + (0 × 2⁶) + (0 × 2⁵) + (1 × 2⁴) + (1 × 2³) + (0 × 2²) + (1 × 2¹) + (0 × 2⁰) = 26 MPL074</p>	
TAP16	") code for paper tape reader/puncher for EIA	<p>This parameter is used to set a hole-punching pattern for the character code ")" onto a paper tape reader/puncher using EIA. Set an eight-digit binary number in decimal form.</p> <p>Example :</p> <p>(0 × 2⁷) + (1 × 2⁶) + (0 × 2⁵) + (0 × 2⁴) + (1 × 2³) + (0 × 2²) + (1 × 2¹) + (0 × 2⁰) = 74 MPL075</p>	

Classification	DATA I/O	Display title	TAPE
Address	Meaning	Description	
TAP24	Tape operation port selection Program type M, E Conditions At I/O startup Unit — Setting range 0, 1	Parameter to select the tape operation ports. If 1 is set, ports COM3 and COM4 can be used by means of a USB-RS232C conversion connector. 0: Tape operation port selection COM1, COM2 1: Tape operation port selection COM1, COM2, COM3, COM4	
TAP25	Paper tape puncher parity-V check Program type M, E Conditions At I/O startup Unit Bit Setting range Binary, eight digits	 <p>7 6 5 4 3 2 1 0</p> <p>0: No parity-V check during paper tape reading 1: Parity-V check during paper tape reading</p>	
TAP26 (bit 0)	Data transfer of a paper tape program which has been punched by M2 Program type E Conditions At I/O startup Unit Bit Setting range 0, 1	0: Tape loading impossible 1: Tape loading possible	
TAP26 (bit 1)	Types of data to be punched onto paper tape during all punching Program type E Conditions At I/O startup Unit Bit Setting range 0, 1	0: All programs and various NC data 1: All programs only	

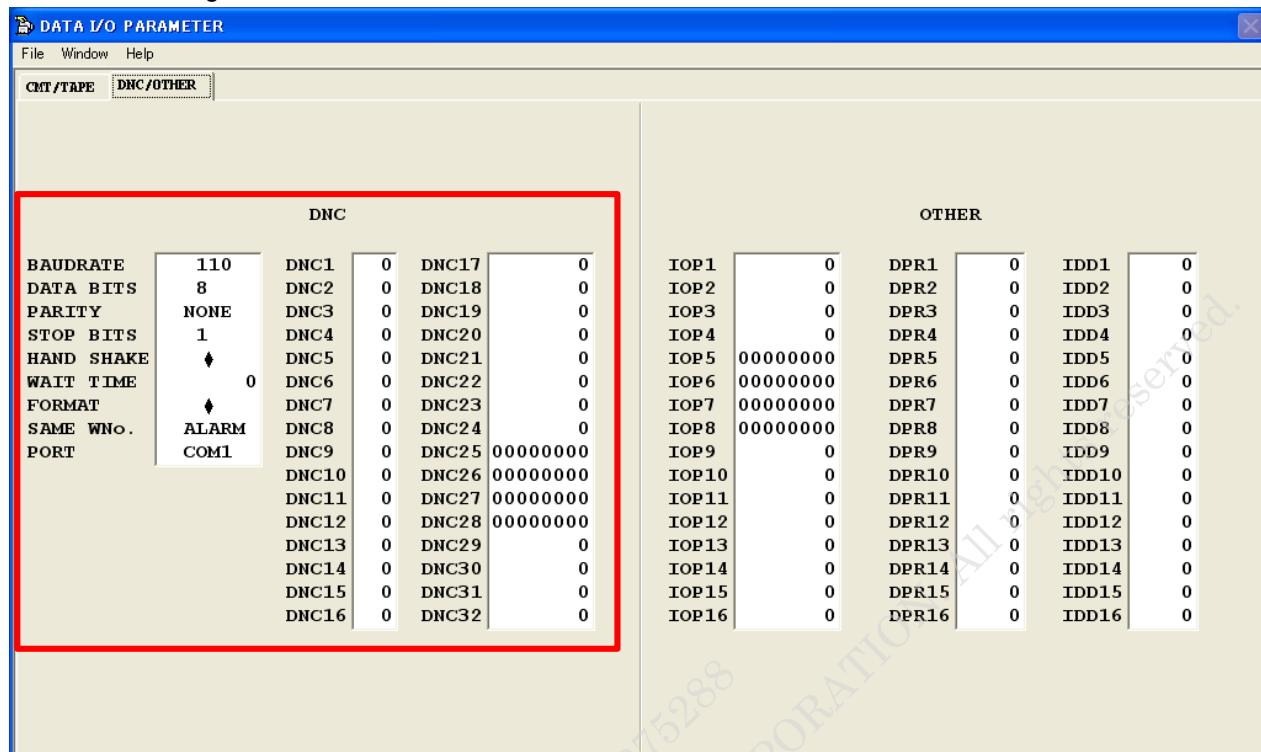
Classification	DATA I/O	Display title	TAPE
Address	Meaning		Description
TAP26 (bit 2)	Input/output of the program name during paper tape reading/punching	Program type Conditions Unit Setting range	0: Valid 1: Invalid
TAP26 (bit 3)	Number of digits of work No. output	Program type Conditions Unit Setting range	0: 8 digits 1: 4 digits
TAP26 (bit 4)	Input/output of material data during punching/reading	Program type Conditions Unit Setting range	0: Output of ASCII data in hexadecimal notation 1: Output in characters

Classification	DATA I/O	Display title	TAPE
Address	Meaning		Description
TAP27	Bit parameter related to program end code (M) for paper tape reader		
	Program type	E	
	Conditions	At I/O startup	
	Unit	Bit	
	Setting range	Binary, eight digits	
TAP29	Number of characters in feed section for paper tape puncher		<p>The number of characters in NULL (feed) that are to be punched at the beginning and end of paper tape</p> <p>Example :</p> <p>MPL078</p>
	Program type	E	
	Conditions	At I/O startup	
	Unit	Character	
	Setting range	0 to 65535	
TAP30	Number of characters in the space between O-number and program for paper tape puncher		<p>The total number of space-characters that are punched out between O-number and program section.</p> <p>MPL079</p>
	Program type	E	
	Conditions	At I/O startup	
	Unit	Character	
	Setting range	0 to 65535	

Classification	DATA I/O	Display title	TAPE
Address	Meaning	Description	
TAP31	Number of characters in the space between programs for paper tape puncher		The total number of space-characters that are punched out between programs when more than one program are punched onto paper tape.
			MPL080
	Program type	E	
	Conditions	At I/O startup	
	Unit	Character	
Setting range		0 to 65535	

2-3-19 Data I/O parameter DNC parameter (DNC)

Parameter setting



D740H0003E

Classification	DATA I/O	Display title	DNC								
Meaning			Description								
BAUDRATE			Baud rate for RS-232C interface Set values								
<table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>110</td><td>4800</td></tr> <tr><td>300</td><td>9600</td></tr> <tr><td>1200</td><td>19200</td></tr> <tr><td>2400</td><td></td></tr> </table>			110	4800	300	9600	1200	19200	2400		
110	4800										
300	9600										
1200	19200										
2400											
DATA BITS			Number of data bits (parameter for RS-232C interface initialization) Set values								
<table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>7</td></tr> <tr><td>8</td></tr> </table>			7	8							
7											
8											
Program type			E								
Conditions			At I/O startup								
Unit			—								
Setting range			110 to 19200								
Program type			E								
Conditions			At I/O startup								
Unit			—								
Setting range			7, 8								

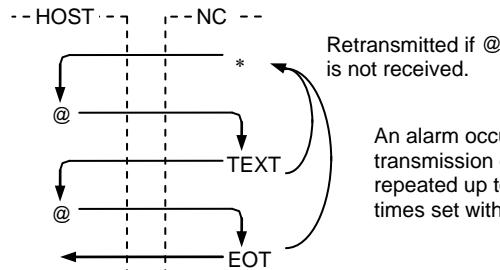
Classification	DATA I/O	Display title	DNC						
Meaning			Description						
PARITY			Parity check (parameter for RS-232C interface initialization) Set values <table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>NONE</td></tr> <tr><td>ODD</td></tr> <tr><td>EVEN</td></tr> </table>	NONE	ODD	EVEN			
NONE									
ODD									
EVEN									
Program type	E								
Conditions	At I/O startup								
Unit	—								
Setting range	—								
STOP BITS			Number of stop bits (parameter for RS-232C interface initialization) Set values <table border="1" style="margin-left: auto; margin-right: auto;"> <tr><td>1</td></tr> <tr><td>1.5</td></tr> <tr><td>2</td></tr> </table>	1	1.5	2			
1									
1.5									
2									
Program type	E								
Conditions	At I/O startup								
Unit	—								
Setting range	—								
WAIT TIME			The waiting time for replies from the connected device during inputting or outputting. An alarm occurs if this time elapses following the final reply.						
Program type	E								
Conditions	At I/O startup								
Unit	0.1 s								
Setting range	0 to 65535								
SAME WNo.			Type of processing to be executed if the machining program of an existing work number is to be loaded <table border="1" style="margin-left: auto; margin-right: auto;"> <thead> <tr> <th>Set values</th> <th>Description</th> </tr> </thead> <tbody> <tr> <td>ALARM</td> <td>Issues an alarm if the received work number already exists.</td> </tr> <tr> <td>LOAD</td> <td>Overrides the program if the received work number already exists.</td> </tr> </tbody> </table>	Set values	Description	ALARM	Issues an alarm if the received work number already exists.	LOAD	Overrides the program if the received work number already exists.
Set values	Description								
ALARM	Issues an alarm if the received work number already exists.								
LOAD	Overrides the program if the received work number already exists.								
Program type	E								
Conditions	At I/O startup								
Unit	—								
Setting range	—								

Classification	DATA I/O	Display title	DNC										
Meaning		Description											
PORT		DNC port selection											
		<table border="1"><thead><tr><th>Set values</th><th>Description</th></tr></thead><tbody><tr><td>COM1</td><td>CF22 serial ch3</td></tr><tr><td>COM2</td><td>CF22 serial ch4</td></tr><tr><td>COM3</td><td>Invalid</td></tr><tr><td>COM4</td><td>Invalid</td></tr></tbody></table>		Set values	Description	COM1	CF22 serial ch3	COM2	CF22 serial ch4	COM3	Invalid	COM4	Invalid
Set values	Description												
COM1	CF22 serial ch3												
COM2	CF22 serial ch4												
COM3	Invalid												
COM4	Invalid												
Program type	E												
Conditions	At I/O startup												
Unit	—												
Setting range	—												

Classification	DATA I/O	Display title	DNC																																						
Address	Meaning	Description																																							
DNC1	Type of terminator		<table border="1"> <thead> <tr> <th>Set values</th><th>Terminator</th></tr> </thead> <tbody> <tr><td>0</td><td>Without terminator</td></tr> <tr><td>1</td><td>EOB or EOR</td></tr> <tr><td>2</td><td>EOB only</td></tr> <tr><td>3</td><td>EOR only</td></tr> <tr><td>4</td><td>One character of your choice</td></tr> <tr><td>5</td><td>Two characters of your choice</td></tr> </tbody> </table>	Set values	Terminator	0	Without terminator	1	EOB or EOR	2	EOB only	3	EOR only	4	One character of your choice	5	Two characters of your choice																								
Set values	Terminator																																								
0	Without terminator																																								
1	EOB or EOR																																								
2	EOB only																																								
3	EOR only																																								
4	One character of your choice																																								
5	Two characters of your choice																																								
Program type	E																																								
Conditions	At I/O startup																																								
Unit	—																																								
Setting range	0 to 5																																								
Terminator code 1																																									
DNC2	Terminator code 1		Effective only when DNC1 is set to 4 or 5.																																						
	Program type	E																																							
	Conditions	At I/O startup																																							
	Unit	—																																							
	Setting range	0 to 255																																							
	Terminator code 2																																								
DNC3	Terminator code 2		Effective only when DNC1 is set to 5.																																						
	Program type	E																																							
	Conditions	At I/O startup																																							
	Unit	—																																							
	Setting range	0 to 255																																							
	DC code parity																																								
DNC5	DC code parity		This parameter is used to specify whether or not a parity bit is to be assigned to the DC code to be output.																																						
	Program type	E																																							
	Conditions	At I/O startup																																							
	Unit	—																																							
	Setting range	0, 1																																							
<table border="1"> <thead> <tr> <th>Set values</th><th>Parity</th><th colspan="8">Hole-punching pattern of DC3 code</th> </tr> </thead> <tbody> <tr><td>0</td><td>No assignment</td><td> </td><td> </td><td> </td><td>●</td><td>●</td><td> </td><td> </td><td>●</td><td>●</td></tr> <tr><td>1</td><td>Assignment</td><td>●</td><td> </td><td> </td><td>●</td><td>●</td><td> </td><td> </td><td>●</td><td>●</td></tr> </tbody> </table>									Set values	Parity	Hole-punching pattern of DC3 code								0	No assignment				●	●			●	●	1	Assignment	●			●	●			●	●	
Set values	Parity	Hole-punching pattern of DC3 code																																							
0	No assignment				●	●			●	●																															
1	Assignment	●			●	●			●	●																															

Classification	DATA I/O	Display title	DNC
Address		Meaning	
DNC9		Number of NC transmission retries during DNC file transfer	
		Program type	M, E
		Conditions	At I/O startup
		Unit	Number of times
		Setting range	0 to 255
DNC10		Number of NC reception retries during DNC file transfer	
		Program type	M, E
		Conditions	At I/O startup
		Unit	Number of times
		Setting range	0 to 255
DNC11		Number of NC transmission/reception retries during DNC command message transfer	
		Program type	M, E
		Conditions	At I/O startup
		Unit	Number of times
		Setting range	0 to 255
DNC12		@ waiting time during DNC transmission	
		Program type	M, E
		Conditions	At I/O startup
		Unit	0.1 s
		Setting range	0 to 255

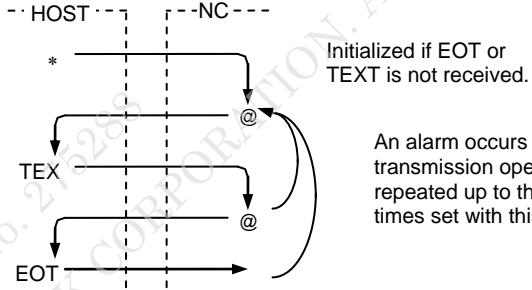
This parameter is used to set the number of times that the * code or TEXT is to be repeatedly transmitted to a host system in case that the @ code is not sent from the host system within the waiting time which has been set at WAIT TIME.



An alarm occurs if the transmission operation is repeated up to the number of times set with this parameter.

MPL081

This parameter is used to set the number of times that the @ code is to be repeatedly transmitted to a host system in the case that the EOT-code or TEXT from the host system is not received within the waiting time which has been set at WAIT TIME.



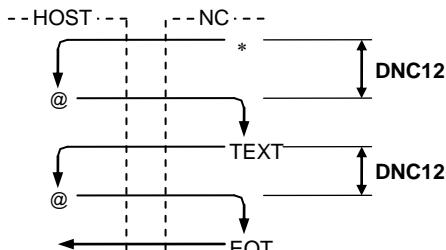
An alarm occurs if the transmission operation is repeated up to the number of times set with this parameter.

MPL082

This parameter is used to set the number of times that transmission/reception of command messages is to be repeated in the case that it is not correctly performed.

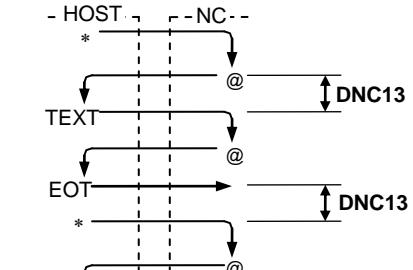
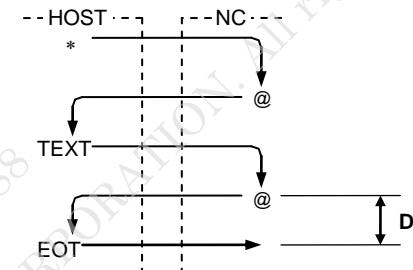
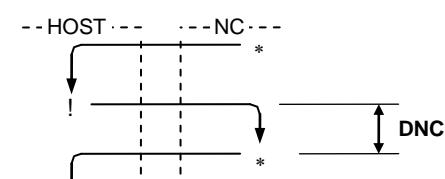
This parameter has almost the same meaning as that of parameters **DNC9** and **DNC10**, except that command messages are interchanged in the case of **DNC11** and files are interchanged in the case of **DNC9** and **DNC10**.

The NC waiting time from transmission of * or TEXT to reception of @ from the host system.

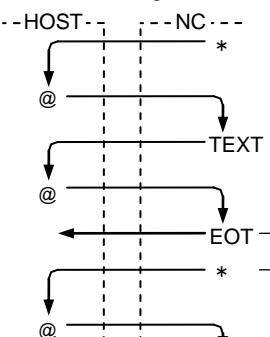
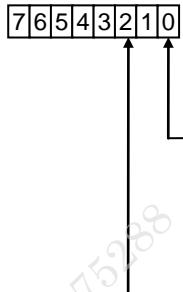
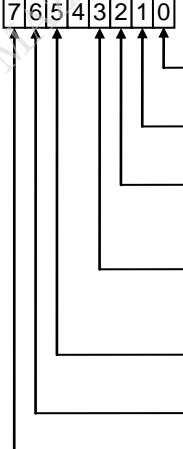


MPL083

(⇒ DNC9)

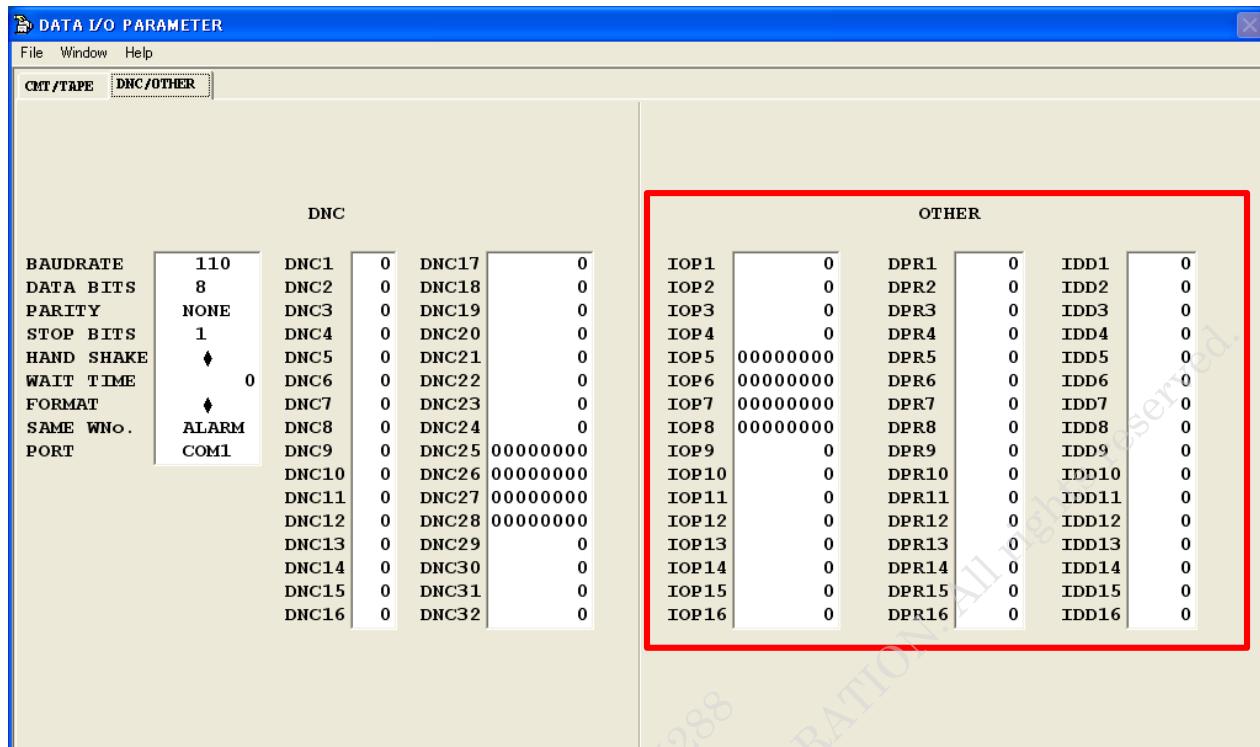
Classification	DATA I/O	Display title	DNC
Address	Meaning	Description	
DNC13	"*", TEXT waiting time during DNC transmission	<p>The NC waiting time from transmission of @ or reception of EOT to reception of * or TEXT from the host system.</p>  <p>MPL084 (⇒ DNC10)</p>	
DNC14	EOT waiting time during DNC transmission	<p>The NC waiting time from transmission of @ to reception of EOT from the host system.</p>  <p>MPL085 (⇒ DNC10)</p>	
DNC15	NC stop time after reception of !	<p>The NC stop time from reception of ! from the host system to transmission of *.</p>  <p>MPL086 Code * is transmitted to the host system if the time that has been set with DNC15 elapses following reception of !.</p>	
DNC16	NC reset time after digital-out	The time from the moment the NC receives the digital-out command to the moment the NC internally resets this command.	
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	0.1 s	
	Setting range	0 to 255	

Classification	DATA I/O	Display title	DNC
Address	Meaning	Description	
DNC17	<p>NC stop time from reception</p> <p>Program type M, E</p> <p>Conditions At I/O startup</p> <p>Unit 0.01 s</p> <p>Setting range 0 to 255</p>	<p>[For NC transmission] The NC stop time from reception of @ from the host system to transmission of EOT or TEXT</p> <p>[For NC reception] The NC stop time from reception of * or TEXT from the host system to transmission of @</p> <p>MPL087</p>	
DNC18	<p>DNC command reply message waiting time</p> <p>Program type M, E</p> <p>Conditions At I/O startup</p> <p>Unit 0.1 s</p> <p>Setting range 0 to 255</p>	<p>The NC waiting time from transmission of command message EOT to reception of command reply message * from the host system.</p> <p>MPL088</p>	
DNC19	<p>DNC machine number</p> <p>Program type M, E</p> <p>Conditions At I/O startup</p> <p>Unit —</p> <p>Setting range 0 to 255</p>	<p>The numbers to be assigned to various machines in order to manage on the host system the tool data, parameters etc. that are specific to the machines being used</p>	
DNC20	<p>NC transmission stop time of DNC (from reception to transmission)</p> <p>Program type M, E</p> <p>Conditions At I/O startup</p> <p>Unit 0.01 s</p> <p>Setting range 0 to 255</p>	<p>The NC stop time from reception of EOT from the host system to transmission of * of the next message</p> <p>MPL089</p>	

Classification	DATA I/O	Display title	DNC
Address	Meaning		Description
DNC21	NC transmission stop time of DNC (from transmission to transmission)		The NC stop time from transmission of EOT to the host system to transmission of * of the next message  MPL090
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	0.01 s	
	Setting range	0 to 255	
DNC25	—		 Select the type of processing to be executed if the tool quantity data within the NC memory mismatches that which has been transferred from the DNC memory. 0 : Issues an alarm if the tool quantity data mismatches. 1 : Executes loading forcibly, even if the tool quantity data mismatches. Handling of tool data and tool files in the M PLUS format 0 : Valid 1 : Invalid
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	Bit	
	Setting range	Binary, eight digits	
DNC26	—		 (1: Valid, 0: Invalid) 1: After program reception, a search is made for the work number of that program. 1: Details of an alarm occurring in DNC are displayed. 1: Loading of programs having the same work number as that of the registered program in NC becomes impossible. 1: The function of the PROGRAM LOCK/ENABLE switch is released. 1: Three digit G-format and G10 format codes input/output for MAZAK data transfer protocol 1: Binary to ASCII format input/output of MAZAK data transfer protocol 1: All programs having work numbers smaller than No. 9000 are erased at the start of program reception. Note : When both bit 5 and bit 6 are set to 1 (enable), this functions for three digit G-format and G10 format codes input/output.
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	Bit	
	Setting range	Binary, eight digits	

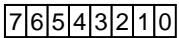
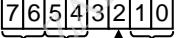
Classification	DATA I/O	Display title	DNC
Address	Meaning		Description
DNC29	Number of retry times with detection of a physical error		
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	Number of times	
	Setting range	0 to 65535	
DNC30	Tool data/tool file message format		Select a tool data/tool file message format. 0: M32 scheme 1: M Plus scheme 2: M640M scheme 3: M640M Pro scheme
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0 to 3	
DNC31	Waiting time for a response from the COM port driver		Set the time interval from the time the response from the COM port driver has disappeared, to the time the initialization of the COM port driver is started.
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	0.1 s	
	Setting range	0 to 65535	

2-3-20 Data I/O parameter OTHER (IOP/DPR/IDD)



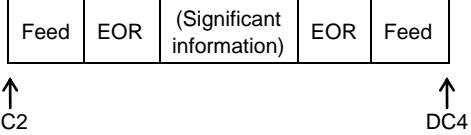
D740H0004E

Classification	DATA I/O	Display title	OTHER
Address	Meaning		Description
IOP5 (bit 1)	<p>Tool data count check during tool data loading</p> <p>Program type M, E</p> <p>Conditions Immediate</p> <p>Unit —</p> <p>Setting range 0, 1</p>		<p>Tool count in tool data is checked when loading tool data in binary format.</p> <p>0: Invalid</p> <p>1: Valid</p> <p>* When tool data does not match the tool count checked during loading, alarm 505 CANNOT LOAD (MISMATCH) occurs.</p>
IOP5 (bit 6)	<p>Overwriting associated with loading of the program of the same work number</p> <p>Program type M, E</p> <p>Conditions Immediate</p> <p>Unit —</p> <p>Setting range 0, 1</p>		<p>Overwriting associated with loading of the same work number as that of the program(s) registered in the NC unit by hard disk, floppy disk, or memory card input/output.</p> <p>0: Alarm without overwriting</p> <p>1: Overwriting</p>

Classification	DATA I/O	Display title	OTHER																																													
Address	Meaning		Description																																													
IOP5 (bit 7)	Overwriting associated with loading of the tool model of the same name		Overwriting associated with loading of the tool model of the same name 0: Next tool model loaded without overwriting 1: Overwriting																																													
	Program type	M, E																																														
	Conditions	Immediate																																														
	Unit	—																																														
	Setting range	0, 1																																														
IOP6	—		 <p>Select between M640T and M640MT for loading MAZATROL programs in text file format. 0: To load MAZATROL programs in text file format created with M640T or T NEXUS 1: To load MAZATROL programs in text file format created with M640MT</p>																																													
	Program type	M																																														
	Conditions	At I/O startup																																														
	Unit	Bit																																														
	Setting range	Binary, eight digits																																														
IOP7	Data entry for communication with the magazine-side display unit		 <p>Specify the serial port number of the NC unit that is to be used for communication with the magazine-side display unit.</p> <table border="1"> <tr><th>Bit 1</th><th>Bit 0</th><th>Serial port</th></tr> <tr><td>0</td><td>0</td><td>COM1</td></tr> <tr><td>0</td><td>1</td><td>COM2</td></tr> <tr><td>1</td><td>0</td><td>COM3</td></tr> <tr><td>1</td><td>1</td><td>COM4</td></tr> </table> <p>Fixed (0)</p> <p>Specify the type of ID amplifier.</p> <table border="1"> <tr><th>Bit 5</th><th>Bit 4</th><th>Type</th></tr> <tr><td>0</td><td>0</td><td>EUCHNER</td></tr> <tr><td>0</td><td>1</td><td>Invalid</td></tr> <tr><td>1</td><td>0</td><td>Invalid</td></tr> <tr><td>1</td><td>1</td><td>Invalid</td></tr> </table> <p>Specify whether the timing of the R-register is to be made valid during tool removal or mounting.</p> <table border="1"> <tr><th>Bit 7</th><th>Bit 6</th><th>Setting</th></tr> <tr><td>0</td><td>0</td><td>Not used</td></tr> <tr><td>0</td><td>1</td><td>Used</td></tr> <tr><td>1</td><td>0</td><td>Invalid</td></tr> <tr><td>1</td><td>1</td><td>Invalid</td></tr> </table>	Bit 1	Bit 0	Serial port	0	0	COM1	0	1	COM2	1	0	COM3	1	1	COM4	Bit 5	Bit 4	Type	0	0	EUCHNER	0	1	Invalid	1	0	Invalid	1	1	Invalid	Bit 7	Bit 6	Setting	0	0	Not used	0	1	Used	1	0	Invalid	1	1	Invalid
Bit 1	Bit 0	Serial port																																														
0	0	COM1																																														
0	1	COM2																																														
1	0	COM3																																														
1	1	COM4																																														
Bit 5	Bit 4	Type																																														
0	0	EUCHNER																																														
0	1	Invalid																																														
1	0	Invalid																																														
1	1	Invalid																																														
Bit 7	Bit 6	Setting																																														
0	0	Not used																																														
0	1	Used																																														
1	0	Invalid																																														
1	1	Invalid																																														
Program type	M, E																																															
Conditions	At power on																																															
Unit	Bit																																															
Setting range	Binary, eight digits																																															

Classification	DATA I/O	Display title	OTHER
Address	Meaning		Description
IOP8 (bit 0)	(Magazine-side display unit IN/OUT) Tool count/time data clear setting		0: Do not clear 1: Clear
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
IOP8 (bit 1)	(Magazine-side display unit IN/OUT) Tool life count/time data clear setting		0: Do not clear 1: Clear
	Program type	M, E	
	Conditions	Immediate	
	Unit	Bit	
	Setting range	0, 1	
IOP9	—		Fixed value (0)
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	
IOP14	MAZA-CARE function (for the Japanese market only)		0: Invalid 1: MAZA-CARE 2 enabled 2: MAZA-CARE 3 enabled * When the term of MAZACARE contract has expired, the function cannot be used even if the parameter is made valid.
	Program type	M	
	Conditions	Immediate	
	Unit	—	
	Setting range	0, 1, 2	

Classification	DATA I/O	Display title	OTHER
Address	Meaning	Description	
DPR1	Baud rate	Baud rate for RS-232C interface	
	110	4800	Set values Baud rate
	300	9600	0 110 4 4800
	1200	19200	1 300 5 9600
	2400		2 1200 6 19200
	Setting range	0 to 6	3 2400
DPR2	Stop bit	Number of stop bits (parameter for RS-232C interface initialization)	
	1	Set values Stop bit	0 1
	1.5		1 1.5
	2		2 2
	Program type	M, E	
	Conditions	At I/O startup	
DPR4	Unit	—	
	Setting range	0 to 2	
	Data bit	Number of data bits (parameter for RS-232C interface initialization)	
	Program type	M, E	Set values
	Conditions	At I/O startup	Set values Data bit
DPR8	Unit	—	0 8
	Setting range	0, 1	1 7
	ISO code CR output and the output file size		This parameter specifies whether "CR" is to be inserted at the beginning of LF (block delimiter) in ISO code output when the output destination is a serial port (DPR14 = 0 or 1).
	Program type	M, E	0: No "CR" insertion 1: "CR" insertion
	Conditions	At I/O startup	This parameter specifies the maximum permissible file size to be used for hard disk output using an external output command macro when the output destination is the hard disk (DPR14 = 4).
	Unit	—	Maximum permissible file size: Entered value × 100K bytes
	Setting range	0 to 255	However, this file size is 100K bytes if the entered value is 0.

Classification	DATA I/O	Display title	OTHER																																	
Address	Meaning	Description																																		
DPR9	Method of handshaking		This parameter is used to select the method of handshaking to control the state of data transfer between the NC system and connected device.																																	
	Program type	M, E	<table border="1"> <thead> <tr> <th>Set values</th> <th>Method</th> <th>Description</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>None</td> <td>No control</td> </tr> <tr> <td>1</td> <td>DC control</td> <td>Complies with control code DC1 through DC4</td> </tr> <tr> <td>2</td> <td>RTS/CTS</td> <td>Complies with device connection RTS/CTS.</td> </tr> </tbody> </table>	Set values	Method	Description	0	None	No control	1	DC control	Complies with control code DC1 through DC4	2	RTS/CTS	Complies with device connection RTS/CTS.																					
Set values	Method	Description																																		
0	None	No control																																		
1	DC control	Complies with control code DC1 through DC4																																		
2	RTS/CTS	Complies with device connection RTS/CTS.																																		
Conditions	At I/O startup																																			
Unit	—																																			
Setting range	0 to 2																																			
DPR10	DC code parity		This parameter is used to specify whether or not a parity bit is to be assigned to the DC code to be output.																																	
	Program type	M, E	<table border="1"> <thead> <tr> <th>Set values</th> <th>Parity</th> <th colspan="8">Hole-punching pattern of DC3 code</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>No assignment</td> <td>●</td><td>●</td><td>●</td><td>●</td><td>●</td><td>●</td><td>●</td><td>●</td><td>●</td><td>●</td> </tr> <tr> <td>1</td> <td>Assignment</td> <td>●</td><td>●</td><td>●</td><td>●</td><td>●</td><td>●</td><td>●</td><td>●</td><td>●</td><td>●</td> </tr> </tbody> </table> <p>Note : This parameter is valid only when the handshaking method is set to DC control (DPR9 is set to 1).</p>	Set values	Parity	Hole-punching pattern of DC3 code								0	No assignment	●	●	●	●	●	●	●	●	●	●	1	Assignment	●	●	●	●	●	●	●	●	●
Set values	Parity	Hole-punching pattern of DC3 code																																		
0	No assignment	●	●	●	●	●	●	●	●	●	●																									
1	Assignment	●	●	●	●	●	●	●	●	●	●																									
Conditions	At I/O startup																																			
Unit	—																																			
Setting range	0, 1																																			
DPR11	Feed section DC code output		Select whether or not DC2 and DC4 codes are to be output to the feed sections.																																	
	Example : 																																			
	Program type	M, E	<table border="1"> <thead> <tr> <th>Set values</th> <th>Description</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>Neither DC2 nor DC4 is output.</td> </tr> <tr> <td>1</td> <td>Only DC2 is output.</td> </tr> <tr> <td>2</td> <td>Only DC4 is output.</td> </tr> <tr> <td>3</td> <td>Both DC2 and DC4 are output.</td> </tr> </tbody> </table>	Set values	Description	0	Neither DC2 nor DC4 is output.	1	Only DC2 is output.	2	Only DC4 is output.	3	Both DC2 and DC4 are output.																							
Set values	Description																																			
0	Neither DC2 nor DC4 is output.																																			
1	Only DC2 is output.																																			
2	Only DC4 is output.																																			
3	Both DC2 and DC4 are output.																																			
Conditions	At I/O startup																																			
Unit	—																																			
Setting range	0 to 3																																			
<p>Note : This parameter is valid only when the handshaking method is set to DC control (DPR9 is set to 1).</p>																																				

Classification	DATA I/O	Display title	OTHER														
Address	Meaning	Description															
DPR12	Waiting time	The waiting time for replies from the connected device during inputting or outputting. An alarm occurs if this time elapses following the final reply.															
	Program type	M, E															
	Conditions	At I/O startup															
	Unit	0.1 s															
	Setting range	0 to 65535															
DPR13	Output format	Selection of output code															
	Program type	M, E	<table border="1"> <thead> <tr> <th>Set values</th> <th>Format</th> <th>Description</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>ISO</td> <td>Output in ISO code</td> </tr> <tr> <td>1</td> <td>EIA</td> <td>Output in EIA code</td> </tr> </tbody> </table>	Set values	Format	Description	0	ISO	Output in ISO code	1	EIA	Output in EIA code					
Set values	Format	Description															
0	ISO	Output in ISO code															
1	EIA	Output in EIA code															
Conditions	At I/O startup																
Unit	—																
Setting range	0, 1																
DPR14	Selection of an output destination port	Port selection															
	Program type	M, E	<table border="1"> <thead> <tr> <th>Set values</th> <th>Port</th> <th>Description</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>COM1</td> <td>CF22 serial ch3</td> </tr> <tr> <td>1</td> <td>COM2</td> <td>CF22 serial ch4</td> </tr> <tr> <td>2</td> <td>COM3</td> <td>USB-RS232C conversion connector</td> </tr> <tr> <td>3</td> <td>COM4</td> <td>USB-RS232C conversion connector</td> </tr> </tbody> </table>	Set values	Port	Description	0	COM1	CF22 serial ch3	1	COM2	CF22 serial ch4	2	COM3	USB-RS232C conversion connector	3	COM4
Set values	Port	Description															
0	COM1	CF22 serial ch3															
1	COM2	CF22 serial ch4															
2	COM3	USB-RS232C conversion connector															
3	COM4	USB-RS232C conversion connector															
Conditions	At I/O startup																
Unit	—																
Setting range	0 to 4																
DPR15	Number of characters or the number of lines in feed section	This parameter denotes the number of characters in NULL (feed) when the output destination is a serial port (DPR14 = 0 or 1).															
	Program type	M, E	<p>Example :</p> <table style="margin-left: auto; margin-right: auto;"> <tr> <td style="text-align: center;">DPR15 characters</td> <td style="text-align: center;">DPR15 characters</td> </tr> <tr> <td style="border: 1px solid black; padding: 2px;">Feed</td> <td style="border: 1px solid black; padding: 2px;">EOR</td> <td style="border: 1px solid black; padding: 2px;">(Significant information)</td> <td style="border: 1px solid black; padding: 2px;">EOR</td> <td style="border: 1px solid black; padding: 2px;">Feed</td> </tr> </table>	DPR15 characters	DPR15 characters	Feed	EOR	(Significant information)	EOR	Feed							
DPR15 characters	DPR15 characters																
Feed	EOR	(Significant information)	EOR	Feed													
Conditions	At I/O startup																
Unit	Character																
Setting range	0 to 65535																
		This parameter denotes the number of lines fed when the output destination is a hard disk (DPR14 = 4).															

Classification	DATA I/O	Display title	OTHER
Address	Meaning	Description	
DPR16	DPRNT operation port selection	Parameter to select the DPRNT operation ports. If 1 is set, ports COM3 and COM4 can be used by means of a USB-RS232C conversion connector. 0: DPR14 port selection 0: COM1, 1: COM2, 4: HDD 1: DPR14 port selection 0: COM1, 1: COM2, 2: COM3, 3: COM4, 4: HDD	
	Program type	M, E	
	Conditions	At I/O startup	
	Unit	—	
	Setting range	0, 1	
IDD1 to IDD16	—	Invalid	
	Program type	—	
	Conditions	—	
	Unit	—	
	Setting range	—	

2

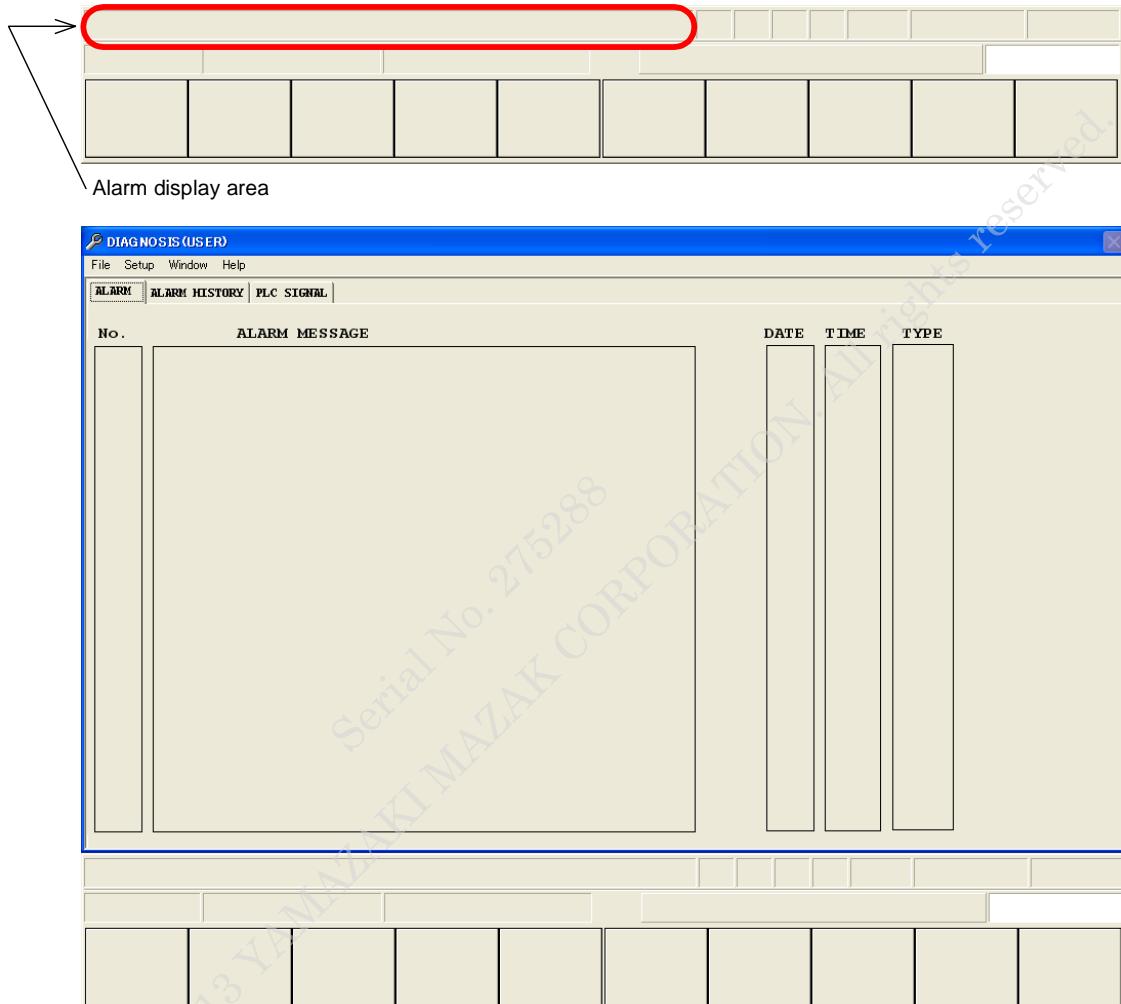
PARAMETER



Copyright(c) 2013 YAMAZAKI MAZAK CORPORATION. All rights reserved.
Serial No. 275288

3 ALARM

If machine failures occur or if erroneous operations are carried out, appropriate alarm numbers and messages will be displayed in the alarm display section of the screen. If alarm display appears, refer to the alarm list to locate and eliminate the cause of the alarm. More than one alarm may be raised at once, depending on the particular status of alarm occurring. In the event of alarm display, therefore, it is highly recommended that the operator should call the **DIAGNOSIS (USER) - ALARM** display on the screen and make sure of the type of alarm.



DIAGNOSIS (USER) - ALARM display

3-1 Outline

1. Scope of this chapter

This chapter describes all the alarms displayed on the screen of NC unit. Always refer to this chapter for eliminating an alarm.

2. Precautions on this chapter

This chapter also lists alarms related to machine model-dependent functions and optional functions. These alarms may therefore include ones not displayed for your machine. Check the type of machine purchased by you and its specifications before you read this chapter.

Note 1: The contents of this list are subjected to change without notice, for NC unit or machine improvement.

Note 2: Any questions about the contents of this list should be communicated to Mazak Technical Center or Technology Center.

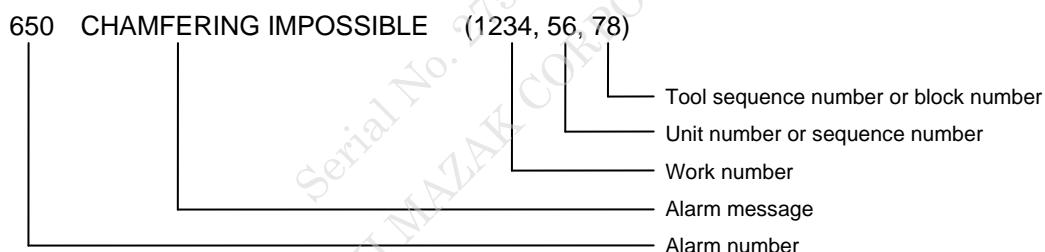
3-1-1 Alarm display

1. Machine-status indicator lamps

In the event of alarm, the machine-status indicator lamp ?ALARM on the operation panel will light up.

2. Display on the screen of NC unit

An alarm will be displayed on the **DIAGNOSIS (USER) - ALARM** display in the following format:



For the **DIAGNOSIS (USER) - ALARM** display, refer to Part 3 OPERATING NC UNIT AND PREPARATION FOR AUTOMATIC OPERATION, 10-1 DIAGNOSIS (USER) - ALARM Display of the Operating Manual.

3. Color of alarm display and its elimination

Alarm display is presented in one of the following colors. Eliminate the alarm according to its display color:

Display color	Alarm elimination
Red	Press the reset key.
Blue	Press the clear key.
Yellow	Press the reset key or clear key.
White	Press the clear key.

3-1-2 Precautions

1. If program-related alarm display appears, that portion of the program in which the alarm has occurred will be displayed within the parentheses next to the alarm message. The meaning of each code in parentheses on the alarm list is listed in the table below.

Code	Meaning
WNo.	Work number (MAZATROL or EIA/ISO)
UNo.	Unit number (MAZATROL)
SNo.	Tool sequence number (MAZATROL)
NNo.	Sequence number (EIA/ISO)
BNo.	Block number (EIA/ISO)
blank	No display, or intra-system alarm processing code

2. The stopped status, clearing procedure, and display color for some types of alarm depend on whether the alarm-encountered program is on the foreground (program selected on the **POSITION** display) or on the background (program selected on the **PROGRAM** display). The above mentioned three types of information for the latter case are indicated with parentheses in the alarm list.
3. The table for an alarm which does not exist remains blank.
4. An alarm may not be displayed for certain machine models or versions of NC-software.

3-2 Detailed Description

3-2-1 Structure of the alarm list

This alarm list is written in the following format:

No.	Message	Type of error	Stopped status	Clearing procedure	Display
[1]	[2] (, ,)	[3]	[4]	[5]	
Cause	[6]				
Action	[7]				

[1] Alarm number

[2] Alarm message

[3] Type of error

Code	Type	Description
A	Operation	A wrong key has been pressed. Or the machine has been operated incorrectly.
B	Registered data	The program or tool data includes an error(s).
C	Servo	Malfunctioning of the servo control mechanism
D	Spindle	Malfunctioning of the spindle control mechanism
E	NC equipment	System (hardware/software) error
F	Machine (PLC)	Machine failure
G	External I/O unit	Malfunctioning of external I/O unit

[4] Stopped status

Code	Status
H	Emergency stop
I	Reset stop
J	Single-block stop
K	Feed stop (hold)
L	Operation continued

[5] Clearing procedure

Code	Procedure
M	Power off → Eliminate cause → Power back on
N	Eliminate cause → Power off → Power back on
O	Eliminate cause → Press reset key
P	Press reset key
Q	Eliminate cause → Press clear key
S	Press clear key

[6] Cause of alarm

[7] Action to be taken to eliminate the cause.

Note : The list for alarms related to PLC machine control (No. 200 to 399 and No. 1200 to 1399) may have a different format.

3-2-2 No. 1 - No. 99, No. 1000 - No. 1099 (System/Drive error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1	EMERGENCY STOP (, ,)				
Cause	—				
Action	—				
2	EMERGENCY STOP (, ,)	E	H	M	Red
Cause	Trouble has occurred in the hardware.				
Action	Turn power off and then back on. If this does not clear the alarm status, contact Mazak Technical Center or Technology Center.				
3	EMERGENCY STOP (, ,)	A	H	M	Red
Cause	The emergency stop button on the operating panel has been pressed.				
Action	Release the pressed state of the emergency stop button and reset the NC unit to its initial state.				
4	— (, ,)				
Cause	—				
Action	—				
5	SYSTEM SOFTWARE ERROR (, ,)	E	H	M	Red
Cause	The contents of the system software and/or custom software have been destroyed.				
Action	Contact Mazak Technical Center or Technology Center.				
6	REMOTE I/O ERROR (, ,)	E	H	M	Red
Cause	—				
Action	Contact Mazak Technical Center or Technology Center.				
7	SRAM MALFUNCTION (, ,)	E	H	M	Red
Cause	The S-RAM mounted on the CPU card has become abnormal.				
Action	Contact Mazak Technical Center or Technology Center.				
8	RAM MALFUNCTION (, ,)	E	H	M	Red
Cause	The RAM mounted on the CPU card has become abnormal.				
Action	Contact Mazak Technical Center or Technology Center.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
9	ABSOLUTE POSITION MALFUNCTION (Alarm No., Axis,)	E	H	M	Red
Cause	The absolute position detection system has lost absolute position data.				
Action	Contact Mazak Technical Center or Technology Center.				
10	DETECTOR MALFUNCTION (Alarm No., Axis,)	E	H	M	Red
Cause	The absolute position detection system has detected its detector error(s).				
Action	Contact Mazak Technical Center or Technology Center.				
11	POSITION REFERENCE MALFUNCTION (Alarm No., Axis,)	E	H	M	Red
Cause	The absolute position detection system has detected an error(s) by cross-checking the absolute position of its detector and the internal coordinate data of the NC unit.				
Action	Contact Mazak Technical Center or Technology Center.				
12	SERVO BATTERY WARNING (Alarm No., Axis,)	E	L	M	Blue
Cause	The battery for retaining absolute position data is running low with a voltage drop, or has exhausted. * Depending on the machine, the message ABSOLUTE POSITION WARNING may be displayed in red.				
Action	The battery must be re-charged or replaced. For the battery information, refer to the Maintenance Manual.				
13	PRE-PROCESSOR MALFUNCTION (, ,)	E	H	M	Red
Cause	The software is not correctly working.				
Action	Contact Mazak Technical Center or Technology Center.				
	(, ,)				
Cause					
Action					
21	SYSTEM ERROR (, ,)	E	H	M	Red
Cause	Software of this system has become abnormal.				
Action	Contact Mazak Technical Center or Technology Center.				
22	AMPLIFIER NOT EQUIPPED (, ,)	E	H	M	Red
Cause	Amplifier power is not yet turned on. Or no signals are transferred yet.				
Action	Check for an incorrectly connected cable, an incorrectly attached connector, an inadequate input supply voltage to the amplifier, an incorrect axis-number switch setting, etc.				
23	— (, ,)				
Cause	—				
Action	—				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
24	— (, ,)				
Cause	—				
Action	—				
25	SAFE OPERATION SYSTEM ALARM (, ,)	E	H	N	Red
Cause	There occurred an alarm in the safe operation system.				
Action	Contact Mazak Technical Center or Technology Center.				
26	SPINDLE SAFE OPER. SYSTEM ALARM (, ,)	E	H	N	Red
Cause	Trouble has occurred for the spindle in the mode of safety-speed operation.				
Action	Contact Mazak Technical Center or Technology Center.				
	(, ,)				
Cause					
Action					
31	SERVO MALFUNCTION 1 (, ,)	C	H	M	Red
Cause	The servo (power-off level) is abnormal.				
Action	Contact Mazak Technical Center or Technology Center.				
32	SERVO PARAMETER MALFUNCTION (, ,)	C	H	M	Red
Cause	The parameters that have been transferred from the NC unit to the servo amplifier during NC power-on are not correct.				
Action	Contact Mazak Technical Center or Technology Center.				
33	SERVO MALFUNCTION 2 (, ,)	C	H	O	Red
Cause	The servo (NC reset level) is abnormal.				
Action	Contact Mazak Technical Center or Technology Center.				
34	SERVO MALFUNCTION 3 (, ,)	C	H	M	Red
Cause	The servo (amplifier power-off level) is abnormal.				
Action	Contact Mazak Technical Center or Technology Center.				
35	OVERLOAD (, ,)	C	H	N	Red
Cause	An extraordinary overload has been detected.				
Action	Check if any collision has occurred on the machine and perform the corresponding recovery, or check and reduce the cutting conditions. If measures other than the above should be taken, contact Mazak Technical Center or Technology Center.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display	
	(, ,)					
Cause						
Action						
41	SPINDLE MALFUNCTION 1 (, ,)	C	H	M	Red	
Cause	The spindle (power-off level) is abnormal.					
Action	Contact Mazak Technical Center or Technology Center.					
42	SPINDLE PARAMETER MALFUNCTION (, ,)	C	H	M	Red	
Cause	The parameters that have been transferred from the NC unit to the spindle amplifier during NC power-on are not correct.					
Action	Contact Mazak Technical Center or Technology Center.					
43	SPINDLE MALFUNCTION 2 (, ,)	C	H	O	Red	
Cause	The spindle (NC reset level) is abnormal.					
Action	Contact Mazak Technical Center or Technology Center.					
44	SPINDLE MALFUNCTION 3 (, ,)	C	H	M	Red	
Cause	The spindle (amplifier power-off level) is abnormal.					
Action	Contact Mazak Technical Center or Technology Center.					
	(, ,)					
Cause						
Action						
51	E2ROM MALFUNCTION (, ,)	E	L	D	Blue	
Cause	Parameters cannot be correctly written into the E2ROM.					
Action	Contact Mazak Technical Center or Technology Center.					
52	BATTERY ALARM (, ,)	E	L	D	Blue	
Cause	The battery provided to retain parameters, machining programs and other types of data within the NC unit has reached the minimum voltage level permissible or has run down.					
Action	It is required that the machining data is rechecked for possible loss or that the battery is recharged or replaced. For battery recharging or replacement, refer to the relevant description given in the Maintenance Manual.					

No.	Message	Type of error	Stopped status	Clearing procedure	Display
53	NC TEMPERATURE WARNING (Note 1, Note 2, Note 3)	E	L	O	Blue
Cause	The temperature of the control unit or operation board has increased above the required level. Note : 1. Location of the temperature alarm 2. Type of temperature alarm 3. On-alarm temperature value				
Action	Reduce the temperature by turning off the NC power or by mounting a cooling unit.				
54	DIO5V MALFUNCTION (, ,)	E	H	O	Blue
Cause	—				
Action	Contact Mazak Technical Center or Technology Center.				
55	REMOTE I/O ERROR (Note 1, Note 2,)	E	H	O	Blue
Cause	—				
Action	Contact Mazak Technical Center or Technology Center. Note : 1. & 00 00 01 00 No.1 system (X0 -, Y0 -) No.2 system (X80 -, Y80 -) No.3 system (X100 -, Y100 -) No.4 system (X280 -, Y300 -) 2. & 00 01 No. 5 system (X380 -, Y400 -) No. 6 system (X400 -, Y700 -) The above example indicates that a communications error has occurred in station 1 of the No. 5 system. 7 6 5 4 3 2 1 0 ----- Station 1 ----- Station 2 ----- : ----- Station 8 ----- The above example indicates that a communications error has occurred in station 1 of the No. 2 system.				
56	SYSTEM SOFTWARE CHECKING (, ,)		H		Red
Cause	The ROMs mounted in the system ROM card are currently being checked for abnormalities.				
Action	Wait for a while. Contact Mazak Technical Center or Technology Center if the alarm is not cleared.				
57	NO PLC (, ,)	F	H	N	Red
Cause	The user PLC is not stored in the NC unit.				
Action	Contact Mazak Technical Center or Technology Center.				
58	CORRUPT PLC (, ,)	F	H	N	Red
Cause	Trouble has occurred with the PLC.				
Action	Contact Mazak Technical Center or Technology Center.				
59	PLC STOPPED (, ,)	F	H	P	Red
Cause	The PLC has ceased running.				
Action	Run the PLC. Contact Mazak Technical Center or Technology Center if the alarm is not cleared.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
	(, ,)				
Cause					
Action					
66	PARAMETER MALFUNCTION (, ,)	E	H	O	Blue
Cause	—				
Action	Contact Mazak Technical Center or Technology Center.				
67	HI-SPEED SYNCTAP AXIS ALARM (, ,)	E	H	O	Blue
Cause	A servo with a system not applicable to high-speed synchronous tapping is connected in spite of the high-speed synchronous tapping option being valid.				
Action	Contact Mazak Technical Center or Technology Center.				
68	HI-SPEED SYNCTAP SPDL ALARM (, ,)	E	H	O	Blue
Cause	A servo with a system not applicable to high-speed synchronous tapping is connected in spite of the high-speed synchronous tapping option being valid.				
Action	Contact Mazak Technical Center or Technology Center.				
69	— (, ,)				
Cause	—				
Action	—				
70	— (, ,)				
Cause	—				
Action	—				
71	ILLEGAL SERVO PARAMETER (, ,)	C	H	M	Blue
Cause					
Action	Contact Mazak Technical Center or Technology Center.				
72	SERVO WARNING (, ,)	E	H	O	Blue
Cause	The servomotor is loaded abnormally.				
Action	Turn off the NC and the machine, remove the cause of the abnormal load, and then turn on the machine and the NC. Contact Mazak Technical Center or Technology Center if the alarm is not cleared.				
	(, ,)				
Cause					
Action					

No.	Message	Type of error	Stopped status	Clearing procedure	Display
81	ILLEGAL SPINDLE PARAMETER (, ,)	E	H	O	Blue
Cause	—				
Action	Contact Mazak Technical Center or Technology Center.				
82	SPINDLE WARNING (, ,)	E	H	O	Blue
Cause	—				
Action	Contact Mazak Technical Center or Technology Center.				
	(, ,)				
Cause					
Action					

3-2-3 No. 100 - No. 199, No. 1100 - No. 1199 (CNC machine control error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display
100	— (, ,)				
Cause	—				
Action	—				
101	SOFT LIMIT (Axis, ,)	A	K	P	Blue
Cause	Some mistake in operations during the automatic operation has caused the tool tip to overstep the area specified in the software limit parameters in the direction of the axis corresponding to the displayed axis name.				
Action	Modify the program so that the tool tip may move within the area specified in the software limit parameters.				
	(, ,)				
Cause					
Action					
113	OVER TRAVEL (Axis, ,)	A	K	P	Red
Cause	The tool tip has reached stroke limit in the direction of the axis corresponding to the displayed axis name.				
Action	Move the tool tip away from the end in manual operation mode.				
	(, ,)				
Cause					
Action					
125	ILLEGAL AXIS EXISTS (, ,)	E	H	O	Red
Cause	During reference-point return, the proximity-point detection limit switch has overrun the position in which the watchdog is mounted.				
Action	Either extend the length of the proximity-point watchdog or reduce the reference-point returning speed. After that, carry out the zero-point returning operation once again.				
126	AXIS HAS NOT PASSED Z PHASE (, ,)	E	H	O	Red
Cause	During initial reference-point return following the power-on action, an axis has not passed through the Z phase of the corresponding detector.				
Action	First actuate the handle for manual pulse feed to move the axis back in the opposite direction to the zero-point, and then carry out the zero-point returning operation once again.				
127	ILLEGAL DIR. FOR ORIGIN RETURN (, ,)	A	K	P	Red
Cause	The axis-movement direction selected with the axis selector button is not correct for the reference-point return in manual operation mode.				
Action	Set the correct direction (+, -) using the axis selector buttons.				
128	OUTSIDE INTERLOCK AXIS (, ,)	A	K	P	Red
Cause	An axis is interlocked because the interlock function has become active (input signal has turned off).				
Action	Clear the active state of the interlock function.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
129	INSIDE INTERLOCK AXIS (, ,)	A	K	P	Red
Cause	The very direction in which the manual skip function has become effective is specified in the axis-movement command. Or the servo-off function is active.				
Action	Deactivate the servo-off function.				
130	NO OPERATION MODE (, ,)	A	K	P	Red
Cause	This message is displayed in the event of incorrect mode selection or a mode selector switch malfunction.				
Action	Contact Mazak Technical Center or Technology Center.				
131	CUTTING FEED OVERRIDE SET AT 0 (, ,)	A	K	P	Blue
Cause	The cutting-feed override value is set to 0 on the machine operating panel.				
Action	Change the cutting-feed override value to one greater than 0. If this alarm message is displayed when the cutting-feed override value is not 0, check the signal line for a short-circuit.				
132	FEEDRATE ZERO (, ,)	A	K	P	Blue
Cause	An attempt has been made to execute an axis movement in the cutting feed mode or dry-run in the automatic operation mode, with the manual feedrate remaining set to 0 on the machine operating panel.				
Action	Change the manual feedrate to a value greater than 0. If this alarm message is displayed when the manual feedrate is not 0, check the signal line for a short-circuit.				
133	STOP SPINDLE (, ,)	D	K	N	Blue
Cause	Spindle rotation did not start when the spindle rotation start command was issued during automatic operation.				
Action	1. Conditions for spindle rotation start are not satisfied. Check the program and tool data. 2. The spindle amplifier and the encoder must be checked for normal operation. Contact Mazak Technical Center or Technology Center.				
134	SPINDLE ROTATION EXCEEDED (, ,)	D	K	N	Red
Cause	The spindle-speed limit has been exceeded.				
Action	Reduce the spindle speed. The spindle amplifier must be checked for normal operation. Contact Mazak Technical Center or Technology Center.				
135	BLOCK START INTERLOCK (, ,)	B	K	N	Red
Cause	The interlock signal to lock the start of the program block has been input.				
Action	The sequence program needs checking for normal functioning. Contact Mazak Technical Center or Technology Center.				
136	CUTTING BLOCK START INTERLOCK (, ,)	B	K	N	Red
Cause	The interlock signal to lock the start of the cutting program block has been input.				
Action	The sequence program needs checking for normal functioning. Contact Mazak Technical Center or Technology Center.				
137	DYNAMIC COMPENSATION EXCEEDED (, ,)	A	K	P	Red
Cause	Dynamic compensation amount exceeded 3 mm (0.12 in).				
Action	Make sure that the workpiece coordinate zero point is centrally positioned in the workpiece, and set the difference between the center of the workpiece and the rotary center of the table to 3 mm (0.12 in) or less.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
138	CANNOT ROTATE TABLE (, ,)	A	K	P	Red
Cause	There are areas where the machining with table rotation cannot be executed.				
Action	Modify the approach point.				
139	PRE-INTERP ACCEL/DECEL ERROR (, ,)	A	K	M	Red
Cause	An abnormality has occurred in the NC unit during the pre-interpolation acceleration/deceleration process.				
Action	Hardware trouble is likely. Contact Mazak Technical Center or Technology Center.				
140	ILLEGAL REFERENCE RETURN No. (, ,)	A	K	P	Blue
Cause	Returning to the second reference point has been commanded in spite of the fact that returning to the first reference point has not yet occurred.				
Action	Return the axis to the first reference point first.				
141	EXCESS SIMULTANEOUS ERROR (, ,)	A	K	P	Blue
Cause	The synchronization error between the master axis and the slave axis during synchronous control has overstepped a predetermined allowable value.				
Action	Move either axis in the direction that the error decreases. Reduce the allowable value to zero (checking invalid), or increase the allowable value.				
142	NONE OR DUPLICATE OPERAT. MODE (, ,)	A	K	P	Blue
Cause	An operation mode has not been selected, or more than one operation mode have been selected. The operation mode selector switch is malfunctioning.				
Action	Check for incorrect wiring of the input mode switch.				
143	ILLEGAL HANDLE FEED AXIS (, ,)	A	K	P	Blue
Cause	A nonexistent axis has been designated as the handle feed axis. Or a handle feed axis has not been designated.				
Action	Check the handle feed axis selection signal line for incorrect wiring. Or check the maximum number of axes that can be used under the current specifications.				

No.	Message (Alarm No., ,)	Type of error	Stopped status	Clearing procedure	Display
144	ILLEGAL CYCLE START It was attempted to start automatic operation under the following conditions:	A	I	O	Red
Cause	Alarm No. Cause				
	0 × 101 The smoothing level is not zero.				
	0 × 102 READY lamp OFF.				
	0 × 103 Reset state not yet cleared.				
	0 × 104 Feed hold is in operation.				
	0 × 105 Hardware overtravel not yet cleared.				
	0 × 106 Software overtravel not yet cleared.				
	0 × 107 No operation mode selected.				
	0 × 108 Two or more operation modes selected.				
	0 × 109 During change in operation modes.				
	0 × 110 During editing on the EIA MONITOR display.				
	0 × 111 During search on the tape data.				
	0 × 113 Abnormality in temperature.				
	0 × 114 Operation restarted in MDI mode.				
	0 × 115 A door opened.				
	0 × 116 During compensation in tandem control.				
	0 × 117 An alarm related to polygonal machining or hob milling not yet cleared.				
	0 × 118 An alarm related to safety barrier not yet cleared.				
Action	Remove the cause, if possible, to (re)start the automatic operation. If not, contact Mazak Technical Center or Technology Center.				
145	REQUIRE ABSOLUTE RECOVERY (Alarm No., ,)	C	K	P	Blue
Cause	The absolute position data has become lost. Trouble has occurred in the absolute position detector.				
Action	Contact Mazak Technical Center or Technology Center.				
146	NOT AUTO MODE (, ,)	A	I	O	Blue
Cause	Automatic operation has been started in a mode other than automatic operation.				
Action	Change the mode to the automatic.				
147	C AXIS TURNING ANGLE OVER (WNo., NNo., BNo.)	A	K	P	Blue
Cause	1. The rotational angle limit at the shaping block connections has been exceeded. 2. The radius of the arc is less than the rotational radius of the C-axis.				
Action	1. Review the program. 2. Review the setting of parameter K1 (rotational radius of the C-axis).				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display																																						
148	CHUCK BARRIER (Note 1, Note 2,)	A (B)	K (L)	O (S)	Red (Blue)																																						
Cause	<p>1. The tool entered in the chuck barrier.</p> <p>Note :</p> <p>1. Interfering component 1</p> <table border="1"> <tr><th>Display</th><td></td></tr> <tr><td>1</td><td>No. 1 spindle chuck</td></tr> <tr><td>2</td><td>No. 2 spindle chuck</td></tr> <tr><td>4</td><td>Tailstock</td></tr> <tr><td>8</td><td>Turret</td></tr> </table> <p>2. Interfering component 2</p> <table border="1"> <tr><th>Display</th><th>Type A</th><th>Type B</th><th>Type C</th></tr> <tr><td>0</td><td>Tool tip</td><td>Turret</td><td>Tool, holder, turret</td></tr> <tr><td>1</td><td>Turret</td><td>Holder</td><td>—</td></tr> <tr><td>2</td><td>—</td><td>Tool</td><td>—</td></tr> </table> <p>Machine type A, B and C above are available according to the settings of the following parameters:</p> <table border="1"> <tr><th>Type</th><th>BA73</th><th>BA129 bit 1</th></tr> <tr><td>A</td><td>Other than 0</td><td>0</td></tr> <tr><td>B</td><td>—</td><td>1</td></tr> <tr><td>C</td><td>0</td><td>0</td></tr> </table> <p>2. Interference is occurring between the tool and chuck in the solid mode.</p>	Display		1	No. 1 spindle chuck	2	No. 2 spindle chuck	4	Tailstock	8	Turret	Display	Type A	Type B	Type C	0	Tool tip	Turret	Tool, holder, turret	1	Turret	Holder	—	2	—	Tool	—	Type	BA73	BA129 bit 1	A	Other than 0	0	B	—	1	C	0	0				
Display																																											
1	No. 1 spindle chuck																																										
2	No. 2 spindle chuck																																										
4	Tailstock																																										
8	Turret																																										
Display	Type A	Type B	Type C																																								
0	Tool tip	Turret	Tool, holder, turret																																								
1	Turret	Holder	—																																								
2	—	Tool	—																																								
Type	BA73	BA129 bit 1																																									
A	Other than 0	0																																									
B	—	1																																									
C	0	0																																									
Action	Review and correct the machining program. If the program is correct, review the tool data and the barrier parameters.																																										
149	TAILSTOCK BARRIER (Note 1, Note 2,)	A (B)	K (L)	O (S)	Red (Blue)																																						
Cause	<p>1. The tool entered in the tail barrier. For details of cause of the alarm, refer to 148 CHUCK BARRIER.</p> <p>2. Interference is occurring between the tool and tailstock in the solid mode.</p>																																										
Action	Review and correct the machining program. If the program is correct, review the tool data and the barrier parameters.																																										
150	WORK PIECE BARRIER (, ,)	A, B	H (L)	O (S)	Red (Blue)																																						
Cause	<p>1. Interference is occurring between the workpiece of the No. 1 turning spindle side and the workpiece of the No. 2 turning spindle side.</p> <p>2. Interference is occurring between the tool and workpiece in the solid mode.</p>																																										
Action	Review and correct the machining program. If the program is correct, review the tool data and the barrier parameters.																																										
151	NO TANDEM CONTROLLING OPTION (, ,)	B	I	N	Red																																						
Cause	Executing the tandem driving function has been attempted despite a tandem driving option not being present.																																										
Action	The tandem driving function can be used only for a special machine. If this alarm occurs in a special machine that allows the use of the tandem driving function, contact Mazak Technical Center or Technology Center.																																										
152	ILLEGAL SYNCHRONIZED AXIS NAME (, ,)	B	I	N	Red																																						
Cause	<p>1. The settings of the address name parameters (M11) on each axis are illegal.</p> <p>2. An attempt has been made to simultaneously control axes other than the master axis that has been set in a parameter.</p>																																										
Action	<p>1. Correct the address names of the master axis and slave axis to the uppercase alphabet and the lowercase alphabet, respectively, and set the address name parameters (M11) of each axis properly.</p> <p>2. Subject the specified master axis to simultaneous control.</p>																																										
153	TRANSFER/PUSH UNFINISHED (, ,)	A	K	O	Red																																						
Cause	<p>1. The axis reached its pressing completion position before a skipping signal was generated.</p> <p>2. The drooping amount during pressing operation cannot overstep the setting in parameter K46.</p>																																										
Action	<p>1. Check whether the pressing completion position in the program is correct.</p> <p>2. Modify the K46 setting as appropriate.</p>																																										

No.	Message	Type of error	Stopped status	Clearing procedure	Display														
154	ZERO RET. NOT ALLOWED (G68 MODE) (, ,)	E	L	S	Blue														
Cause	Zero-point return mode has been selected during three-dimensional coordinate conversion.																		
Action	1. Select an operation mode other than the zero-point return mode. 2. To execute manual zero-point return, cancel three-dimensional coordinate conversion mode.																		
155	BARRIER TURRET/TOOL HEAD (Note 1, Note 2,)	A	K	O	Red														
Cause	The barriers of the upper and lower turrets interferes with each other. For details of cause of the alarm, refer to 148 CHUCK BARRIER .																		
Action	Correct the machining program. If the program is correct, review the parameters related to the barrier.																		
156	ILLEGAL MEASURING SENSOR SIGNAL (, ,)	A, B	H	O	Red														
Cause	The measuring sensor has turned on at a position in a non-measuring area.																		
Action	If the sensor has come to contact with an object, confirm the program. In other cases, confirm the mounting status of the sensor.																		
157	MEASURED RESULT MALFUNCTION (, ,)	A, B	H	O	Red														
Cause	Measurement results are not normal.																		
Action	Check the program. Check if the sensor tool length registered on the TOOL DATA display is correct. Check the mounting status of the sensor.																		
158	ILLEGAL COMMAND CROSS MACHINING (, ,)	A	K	Q	Blue														
Cause	It was attempted to manually control an axis currently under the control for cross machining.																		
Action	The axes under the cross machining control cannot be controlled manually.																		
159	CROSS MACHINING IMPOSSIBLE (Alarm No., ,)	E	I	P	Red														
Cause	<table border="1"> <thead> <tr> <th>Alarm No.</th><th>Cause</th></tr> </thead> <tbody> <tr> <td>0 × 0001</td><td>The specified axis does not exist in the counterpart system.</td></tr> <tr> <td>0 × 0002</td><td>The self-system is specified as the system to be cross-controlled.</td></tr> <tr> <td>0 × 0004</td><td>The cross machining control is specified for an axis which is already under the control in question.</td></tr> <tr> <td>0 × 0010</td><td>A command for cross machining control is given from the counterpart system.</td></tr> <tr> <td>0 × 0040</td><td>In the mode of inclined-axis control a cross-control command is given for the fundamental axis (X-axis) without positioning of the inclined axis (Y-axis) in its zero point. In the mode of inclined-axis control a cross-control command is given for the inclined axis (Y-axis).</td></tr> <tr> <td>0 × 0400</td><td>At the execution of a cross-machining command, the axis movement completion wait time exceeded the time set at parameter SU54.</td></tr> </tbody> </table>					Alarm No.	Cause	0 × 0001	The specified axis does not exist in the counterpart system.	0 × 0002	The self-system is specified as the system to be cross-controlled.	0 × 0004	The cross machining control is specified for an axis which is already under the control in question.	0 × 0010	A command for cross machining control is given from the counterpart system.	0 × 0040	In the mode of inclined-axis control a cross-control command is given for the fundamental axis (X-axis) without positioning of the inclined axis (Y-axis) in its zero point. In the mode of inclined-axis control a cross-control command is given for the inclined axis (Y-axis).	0 × 0400	At the execution of a cross-machining command, the axis movement completion wait time exceeded the time set at parameter SU54 .
Alarm No.	Cause																		
0 × 0001	The specified axis does not exist in the counterpart system.																		
0 × 0002	The self-system is specified as the system to be cross-controlled.																		
0 × 0004	The cross machining control is specified for an axis which is already under the control in question.																		
0 × 0010	A command for cross machining control is given from the counterpart system.																		
0 × 0040	In the mode of inclined-axis control a cross-control command is given for the fundamental axis (X-axis) without positioning of the inclined axis (Y-axis) in its zero point. In the mode of inclined-axis control a cross-control command is given for the inclined axis (Y-axis).																		
0 × 0400	At the execution of a cross-machining command, the axis movement completion wait time exceeded the time set at parameter SU54 .																		
Action	Review and correct the program. Check and correct the value set at parameter SU54 for alarm No. 0 × 0400.																		
160	ILLEGAL NUMBER OF CROSS AXIS (, ,)	E	I	P	Red														
Cause	The number of axes in the remote system has become zero as a result of crossing command execution.																		
Action	Review and correct the program.																		

No.	Message	Type of error	Stopped status	Clearing procedure	Display
161	SUPERPOSITION CTRL IMPOSSIBLE (WNo., UNo., SNo.)	E	I	P	Red
Cause	It was attempted to start superposition control with an unavailable axis. It was attempted to start superposition control under incompatible modal conditions.				
Action	Check the parameters related to the type of acceleration & deceleration and to the motion speed during superposition control.				
162	ILLEGAL SUPERPOSITION PARAMETER (WNo., UNo., SNo.)	E	I	P	Red
Cause	Incorrect parameter settings do not allow the superposition control to be started.				
Action	Check the parameters related to the type of acceleration & deceleration and to the motion speed during superposition control.				
163	BARRIER WORK (WNo., UNo., SNo.)	A	K	O	Red
Cause	There is a danger that one workpiece will interfere with the other. There is a danger that the partition plate will interfere with the workpiece or the lower turret.				
Action	Review the barrier parameters, or for automatic operation, correct the program properly.				
164	ILLEGAL CMD SAFE OPER SYS ALARM (, ,)	A	J	Q	Blue
Cause	With the machine under the control for safety speed, a command for the following was given: Threading, Synchronous tapping, Cross machining, Superposition control.				
Action	Close the door to operate the machine.				
165	CANNOT MAKE SAFE OPER SYS ACTIVE (, ,)	A	L	Q	Blue
Cause	The command signal for the control for safety speed was turned on in the following modes of machining: Threading, Synchronous tapping, Cross machining, Superposition control.				
Action	Close the door to operate the machine.				
166	LOWER TURRET ALREADY ASSIGNED (, ,)	A	K	O	Red
Cause	An M810 command was given from one HD side to operate the lower turret that was currently used on the other HD side.				
Action	Give the M810 command after the use of the lower turret on the other side.				
167	ILLEGAL OPER TOOL TIP PT CTRL (, ,)	A	K (O)	O (S)	Blue
Cause	An attempt was made in the mode of tool tip point control to perform such an improper operation as follows: 1. Manual interruption 2. Interruption using the manual pulse handle 3. MDI interruption				
Action	Do not attempt to perform any of the above operations 1 to 3 in the mode of tool tip point control.				
168	ILLEGAL OPER 5X RADIUS COMP. (, ,)	A	K (O)	O (S)	Blue
Cause	An attempt was made in the mode of tool radius compensation for five-axis machining to perform such an improper operation as follows: 1. Manual interruption 2. Interruption using the manual pulse handle 3. MDI interruption				
Action	Do not attempt to perform any of the above operations 1 to 3 in the mode of tool radius compensation for five-axis machining.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
169	HIGH SMOOTHING CTR. ILLEGAL OPE (, ,)	A	K (O)	Q (S)	Blue
Cause	In the mode of high-speed smoothing control an unavailable operation (e. g. manual interruption) was attempted.				
Action	Manual interruption cannot be performed in the mode of high-speed smoothing control.				
170	W AXIS THRUST FORCE ALARM (, ,)	E	J	O	Red
Cause	W-axis thrust has not been obtained.				
Action	Review and correct the program.				
171	ILLEGAL W AXIS COMMAND (, ,)	E	J	O	Red
Cause	W-axis operation was programmed during the application of the W-axis thrust.				
Action	Review and correct the program.				
172	B AXIS NOT AVAILABLE (, ,)	B	I	N	Red
Cause	The head (B) axis is used without the B-axis control option.				
Action	Purchase the B-axis control option.				
173	2ND SPDL C AXIS NOT AVIALABLE (, ,)	B	I	N	Red
Cause	The No. 2 spindle/C-axis is used without the No. 2 spindle/C-axis option.				
Action	Purchase the No. 2 spindle/C-axis option.				
174	2ND SPDL 1/1000 N/A (, ,)	B	I	N	Red
Cause	The No. 2 spindle/C-axis is used without the No. 2 spindle high-precision indexing option.				
Action	Purchase the No. 2 spindle high-precision indexing option.				
175	TABLE UNBALANCE CHECK N/A (, ,)	B	I	N	Red
Cause	Vibration detection is used without the rotary table vibration detection option.				
Action	Purchase the rotary table vibration detection option.				
176	CANNOT CHANGE TIME CONSTANT (System No., ,)	B	I	P	Red
Cause	Changing the cutting feed time constant in the system has been attempted during synchronous tapping, threading, or control axis superposition.				
Action	Review and correct the program.				
177	ONE-TOUCH TUNING IMPOSSIBLE (System No., ,)	B	I	P	Red
Cause	The inertia estimation has been attempted by G297 during movement of the axis whose inertia was to be estimated. The G298 parameter setup command has been set during movement of the parameter setup reference axis or of some axis assigned to the parameter data development system. The G298 parameter setup command has been set during cutting feed time constant changeover or control axis superposition. The G298 parameter setup command has been set during axis rotation.				
Action	Review and correct the program.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
178	INERTIA TOO LARGE (System No., ,)	B	I	P	Red
Cause	The estimated inertia has exceeded the maximum inertia specified in the TSVC parameter or the TSPC parameter.				
Action	Correct the settings of the TSVC parameter or the TSPC parameter.				
179	CANNOT AUTO MEASURE INERTIA (System No., ,)	B	I	P	Red
Cause	Resonance has not been reduced during inertia estimation. The minimum inertial is not specified in TSVC or TSPC. The inertia estimation itself has failed.				
Action	Correct the settings of the TSVC parameter or the TSPC parameter.				
180	SYNCHRO TAP ILLEGAL COMMAND (WNo., UNo., SNo.)	B	I	P	Red
Cause	An orienting command has been entered during synchronous tapping.				
Action	Review and correct the program.				
181	NO MDI DATA (System No., ,)	B	L	S	Blue
Cause	Although operation has been started in MDI mode, data has been found to be missing.				
Action	Set data in the MDI window.				
182	CANNOT DETACH AXIS (Axis, UNo., SNo.)	B	L	Q	Red
Cause	The axis removal request signal has been turned on when the axis removal-in-progress signal was on (after turning off the axis removal request signal, the operator has turned the axis removal request signal back on without waiting for the axis removal-in-progress signal to turn off). After the axis removal request signal was turned on for the absolute position control axis, the axis removal request signal has been turned off during display of alarm 71 ILLEGAL SERVO PARAMETER .				
Action	Turn on the axis removal request signal when the axis removal-in-progress signal is off. Clear the alarm.				
183	CANNOT MDI INTERRUPT (, ,)	A	L	Q	Blue
Cause	MDI interruption has been attempted during automatic operation.				
Action	MDI interruption is not possible. Return to original mode and resume operation.				
184	ILLEGAL OPER IN G54.4 MODE (, ,)	A	L	Q	Blue
Cause	An attempt was made in the mode of workpiece placement error correction to perform such an improper operation as follows: 1. Manual interruption 2. Interruption using the manual pulse handle 3. MDI interruption				
Action	Do not attempt to perform any of the above operations 1 to 3 in the mode of workpiece placement error correction.				
185	ILLEGAL OPER IN G68.2 MODE (, ,)	A	L	Q	Blue
Cause	An attempt was made in the mode of inclined-surface machining to perform such an improper operation as follows: 1. Manual interruption 2. Interruption using the manual pulse handle 3. MDI interruption				
Action	Do not attempt to perform any of the above operations 1 to 3 in the mode of inclined-surface machining.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
186	W-THERMAL COMP. OVER LIMIT (, ,)	B	L	S	Blue
Cause	The amount of compensation for workpiece thermal expansion exceeded the effective range of +0.5 to -0.5 mm.				
Action	The operation is continued with the amount of error compensation reduced to the limit (+0.5 to -0.5 mm). Change the reference point for workpiece thermal expansion compensation to a point on the workpiece, or execute the workpiece thermal expansion compensation enable/disable command when the tool tip is in close proximity to the workpiece.				
187	SET W-THERMAL COMP. TOOL LENGTH (, ,)	B	L	S	Blue
Cause	The workpiece thermal expansion compensation command is issued to a tool for which tool data is not entered.				
Action	Enter the tool data (tool length). If the tool length is not specified, compensation will be made on the assumption that the tool tip is at the spindle nose (or at the center of rotation of the B-axis if the machine has a spindle head rotating mechanism).				
188	SET W-THERMAL COMP. REF. POINT (, ,)	B	J	P	Red
Cause	Although the reference point for workpiece thermal expansion compensation was not set, the workpiece thermal expansion compensation command was issued.				
Action	Set the reference point for workpiece thermal expansion compensation before issuing the workpiece thermal expansion compensation command.				
189	ILLEGAL SPINDLE DIRECTION (WNo., UNo.,)	D	K	N	Red
Cause	The directions of rotation of the turning tool selected in the manual program unit and the spindle do not match.				
Action	Check the direction of rotation set for the tool selected in the manual program unit and select a correct tool. Alternatively, set TC144 bit3 to "0" to enable automatic output of the spindle rotation command.				
190	— (, ,)				
Cause	—				
Action	—				
191	FILE SYSTEM I/O ERROR (WNo., UNo., SNo.)	E	I	P	Red
Cause	An internal error(s) has occurred during program data change by the function of VFC, MMS etc.				
Action	After checking the entire data of the program being executed, tool data, tool file, parameters, etc., save the data using the data I/O operation and then contact Mazak Technical Center or Technology Center.				
192	EXECUTION IMPOSSIBLE (WNo., UNo., SNo.)	E	I	P	Red
Cause	An internal error(s) has occurred during execution of the MMS unit.				
Action	After checking the entire data of the program being executed, tool data, tool file, parameters, etc., save the data using the data I/O operation and then contact Mazak Technical Center or Technology Center.				
193	NO TOOL IN MAGAZINE (WNo., UNo., SNo.)	B	I	P	Red
Cause	Tool data that correspond to the pocket numbers being displayed in the "TNo." item of the POSITION display are unregistered.				
Action	Register the tool data.				
194	NO TOOL DATA IN PROGRAM (WNo., UNo., SNo.)	E	I	P	Red
Cause	An internal error(s) has occurred when surface speed or feedrate changing by VFC function was under way.				
Action	After checking the entire data of the program being executed, tool data, tool file, parameters, etc., save the data using the data I/O operation (floppy disk) and then contact Mazak Technical Center or Technology Center.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
195	WRONG MEASURING DIRECTION (, ,)	A	I	O	Red
Cause	During the second or subsequent rounds of manual measurement, an attempt has been made to perform skipping in a direction not available for measurement.				
Action	Perform measurements in the correct direction.				
196	WRONG MEASURING POINT (, ,)	A	I	P	Red
Cause	During the second or subsequent rounds of manual measurement, an attempt has been made to measure an illegal point.				
Action	Measure correct points.				
197	UNREGISTERED HEAD DATA (, ,)	B	I	P	Red
Cause	Head data of the head number being used during MMS, MDI MMS or manual measurement does not exist.				
Action	Contact Mazak Technical Center or Technology Center.				
198	NO HEAD DATA (, ,)	B	I	P	Red
Cause	Head data of the head number being used during MMS, MDI MMS or manual measurement is partly missing.				
Action	Contact Mazak Technical Center or Technology Center.				
199	DIVISION BY ZERO (, ,)		I	P	Red
Cause	An attempt has been made to carry out divisions by zero inside the NC unit during measurement of the degree-of-straightness on the MEASURE display.				
Action	Check the touch sensor for abnormalities. Carry out measurements once again if the touch sensor is normal.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1101	INTERFERE (, ,)	A	K	S	Blue
Cause	Interference occurs between <Interfering section 1> and <Interfering section 2>.				
Action	Press the clear key. When the clear key is pressed, the interference check for the secondary check distance is carried out. When the reset key is pressed, the process returns to the interference check for the primary check distance.				
1102	INTERFERE (, ,)	A	I	P	Red
Cause	Interference occurs between <Interfering section 1> and <Interfering section 2>.				
Action	Press the reset key to clear the alarm. When the reset key is pressed, the process returns to the interference check for the primary check distance. Alternatively, disable the machine interference check to clear the alarm.				
1103	INTERFERE CHECK DATA OVERLOAD (, ,)	B	L	S	Blue
Cause	The 3D interference check process cannot be performed within a proper time.				
Action	Set a simple workpiece model or a fixture model or a tool model.				
1104	RAPID FEED OVERRIDE SET AT 0 (, ,)	A	K	P	Blue
Cause	The rapid feed override value set up on the operating panel is "0".				
Action	Set the override value other than "0".				
1105	SAFETY SHEILD CALCULATING (, ,)	B	L	S	Blue
Cause	Machine operation has been stopped because of a delay in interference check process.				
Action	The machine will be restarted when the interference check process catches up with the operation. If this alarm recurs, set up a simplified material model, fixture model or tool model.				
1106	SAFETY SHIELD FAILURE (Cause, Alarm No.,)	B	L	N	Red
Cause	If the processing load of the PC is too high, it may not be possible to update the model properly. A press of the reset key returns operation to a normal state, thus allowing interference checks to be performed properly. 1. Creation of the model has failed. 2. The interference check process has not been performed properly. 3. An NC-PC communications error (time-out error) has occurred. 4. An illegal access has been detected in processing at the NC side. 5. Preparation of the safety shield has not completed normally because an alarm occurred during preparation of the safety shield. * For causes 1 to 4, the alarm No. given as the argument is not effective.				
Action	Review and correct the parameter and model data. Then press the reset key. If this does not clear the alarm status, turn power off and then back on. If the alarm is not caused by software or hardware fault, pressing the reset key to update the model may clear the alarm. If the alarm cannot be cleared, contact Mazak Technical Center or Technology Center. * For cause 5, take the necessary action according to the corrective action for the alarm number.				
1107	INCOMPLETE TOOL DATA (, ,)	B	L	N	Red
Cause	Manual tool measurement results cannot be calculated because of tool data items remaining blank.				
Action	Confirm the settings of the machining section, rotational direction, and nose R in the tool data items, and enter the correct data.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1108	CANNOT CHANGE OPERATION MODE (, ,)	A	L	Q	Blue
Cause	The memory operation mode key has been pressed during tape operation.				
Action	Changeover to memory operation mode during tape operation is prohibited. Reselect tape operation mode before restarting operation.				
1109	TOOL LENGTH OVER (, ,)	B	L	N	Red
Cause	The tool length input range has been overstepped as a result of tool measurement.				
Action	Use a tool of any tool length value falling within the range defined in parameters SU110–SU113 .				
	(, ,)				
Cause					
Action					
1122	INTERFERE (WNo., UNo., SNo.)	B	L	N	Red
Cause	Machine interference has been detected with the NC axis active in automatic operation mode.				
Action	Press the RESET key. Modify the program to prevent machine interference from recurring.				
1123	ILLEGAL OPE (SAFETY SHIELD) (Cause, ,)	A	I	P	Red
Cause	The operations that make interference checking impossible were performed during automatic operation when the INTELLIGENT SAFETY SHIELD function was valid. The values of "Cause" denote the causes below. 1: After an EIA modal search under the invalid status of the INTELLIGENT SAFETY SHIELD in automatic operation mode, the Automatic Operation INTELLIGENT SAFETY SHIELD function was made valid in an attempt to start automatic operation. 2: Following completion of the EIA modal search, manual operation mode was selected and after the axis was moved to a position different from that before the search, memory mode was reselected and a restart operation was attempted. 3: An attempt was made to start the MDI operation at the same time for both the upper and the lower turret.				
Action	Confirm the cause of the alarm. Also, avoid performing the corresponding operations when the Automatic Operation INTELLIGENT SAFETY SHIELD function is valid. Alternatively, restart operation after making the automatic operation interference checking function invalid with the Automatic Operation INTELLIGENT SAFETY SHIELD menu.				
1124	ILLEGAL CYCLE START (, ,)	B	L	N	Red
Cause	Starting either the hard disk, Ethernet, IC card, or tape operation was attempted when the Automatic Operation INTELLIGENT SAFETY SHIELD function was valid.				
Action	Interference checking during automatic operation is prohibited in the hard disk, Ethernet, IC card, or tape operation mode. Restart operation after making the automatic operation interference checking function invalid with the Automatic Operation INTELLIGENT SAFETY SHIELD menu.				
1125	SAFETY SHIELD CALCULATING (, ,)	A	L	Q	Blue
Cause	Calculation for interference checking during automatic operation is in progress.				
Action	When the Automatic Operation INTELLIGENT SAFETY SHIELD function is valid, the start of automatic operation displays this alarm message, but the alarm will be automatically cleared when the calculation is completed. If this alarm occurs except during the start of automatic operation, change the model to a simple one. If the single block or feed hold signal is input while an alarm is active, the next operation is not executed even when the alarm is cleared and the machine remains stopped until the cycle start button is pressed again.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1126	— (, ,)				
Cause	—				
Action	—				
1127	INVALID SAFETY SHIELD (, ,)	B	L	S	Blue
Cause	<p>The INTELLIGENT SAFETY SHIELD for automatic operation has been canceled.</p> <ol style="list-style-type: none"> With the INTELLIGENT SAFETY SHIELD being valid for automatic operation, a command which is not compatible with interference check was executed. With the INTELLIGENT SAFETY SHIELD being valid for automatic operation, an attempt was made to start a program stored in the storage areas for Hard Disk operation, Ethernet operation, and IC Card operation, or a program prepared for Tape operation. 				
Action	Press the clear key to clear the alarm. Be all the more cautious of allowing interference because the INTELLIGENT SAFETY SHIELD is not working.				
1128	SAFETY SHIELD CONDITION CHANGED (, ,)	A	L	S	Blue
Cause	With the INTELLIGENT SAFETY SHIELD being valid for automatic operation, the feed-overriding value has been changed.				
Action	Press the clear key to clear the alarm. In addition, take care about interference because the actual operation may differ from the result of the interference check due to the change in the feed rate override.				
	(, ,)				
Cause					
Action					
1132	ILLEGAL CHOPPING CONDITION (Cause, ,)	A	I	P	Red
Cause	<ol style="list-style-type: none"> The chopping axis is in the servo-off or removed status. The chopping axis is the rotating axis, basic axis or tilting axis under tilt control, or tandem axis. The top dead point and the bottom dead point are the same. The chopping axis has not been returned to its zero point. An attempt was made to start chopping when the chopping axis is selected for the axis to control manually. The chopping feed rate is zero. An attempt was made to change the chopping axis during chopping. An attempt was made in the replay mode of the fixed offset method to use an offset number of which record has not been written. An M-code command for axis movement is issued during chopping. 				
Action	<p>Press the reset key. Take action below against respective causes described above.</p> <ol style="list-style-type: none"> Ensure that the chopping axis is not in the servo-off or removed status. Select an axis other than the rotating axis, basic axis or tilting axis under tilt control, or tandem axis for the chopping axis. Review the position of the top dead point or bottom dead point. Return the chopping axis to its zero point. Select another axis for an axis to control manually and start chopping. Review the chopping feed rate setting. Do not change the chopping axis during chopping. Use an offset number of which recording is finished in the replay mode of the fixed offset method. Do not issue any M-code command for axis movement during chopping. 				

No.	Message	Type of error	Stopped status	Clearing procedure	Display									
1133	ILLEGAL OPERATION IN CHOPPING (Cause, ,)	A	L	Q	Blue									
Cause	Manual interrupt or zero-point return mode for the chopping axis is selected during chopping.													
Action	Do not attempt to select manual interrupt for the chopping axis during chopping. Do not select the zero-point return mode. Remove the cause and press the clear key.													
	(, ,)													
Cause														
Action														
1137	SAFETY SHIELD CONDITION CHANGED (, ,)	A	L	S	Blue									
Cause	The machine has been operated using a spare tool with the INTELLIGENT SAFETY SHIELD function while automatic operation was enabled.													
Action	Press the clear key. In addition, take care about interference because the actual operation may differ from the result of the interference check due to the difference between the actual tool and the tool model used for interference check.													
1138	INVALID SAFETY SHIELD(MATERIAL) (, ,)	A	L	S	Blue									
Cause	A material interference check cannot be carried out while the modal restarting process of the EIA/ISO program is in progress.													
Action	Check that there is no interference with the material along the path from the current position to the restarting start position. Retract the tool to a safe position before executing modal restart of the EIA/ISO program.													
1139	ILLEGAL COMMAND (SAFETY SHIELD) (, ,)	A	K	S	Red									
Cause	With parameter settings of F156 bit0 = 0 and F156 bit1 = 1, a cutting command that incorporates a rotary axis travel was executed while one of the following interpolation modes was active. <ul style="list-style-type: none"> • With the C-/U-axis Interpolation other than G7.1 (cylindrical interpolation) or G12.1 (polar coordinate interpolation) active • With the B-axis Interpolation active 													
Action	Add the material model cutting invalid M-code in the program before starting the operation. It is also possible to select whether the operation should be continued or stopped by an alarm by using the following parameter settings. <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 15%;"></td><td style="width: 35%;">0</td><td style="width: 50%;">1</td></tr> <tr> <td>F156 bit0</td><td>Continue operation with material cutting enabled only at the end position.</td><td>According to the setting of F156 bit1</td></tr> <tr> <td>F156 bit1</td><td>Continue operation with material cutting disabled.</td><td>Stop with an alarm</td></tr> </table>						0	1	F156 bit0	Continue operation with material cutting enabled only at the end position.	According to the setting of F156 bit1	F156 bit1	Continue operation with material cutting disabled.	Stop with an alarm
	0	1												
F156 bit0	Continue operation with material cutting enabled only at the end position.	According to the setting of F156 bit1												
F156 bit1	Continue operation with material cutting disabled.	Stop with an alarm												
	(, ,)													
Cause														
Action														

3-2-4 No. 200 - No. 399, No. 1200 - No. 1399 (PLC machine control error)

To clear the alarms indicated below, first eliminate the cause of the alarm and press the  or  key.

Detector SQ Nos. that vary depending on the model are given for each model.

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
200 (F0)	HYDRAULIC UNIT PRESSURE DOWN	The hydraulic pressure has dropped continuously for 2 or more seconds.	Feed hold Spindle stop	Chiller unit X5
201 (F1)	CHILLER MALFUNCTION	The hydraulic pressure has dropped continuously for 2 or more seconds.	Stop in the single block operation mode	Hydraulic temperature control unit X4
202 (F2)	AIR PRESSURE DOWN	The air pressure for spindle lubrication has dropped.	Feed hold Spindle stop	SP5 XA2
206 (F6)	SPINDLE LUB. CHILLER MALF.	Alarm related to the spindle lubrication chiller unit	Feed hold Spindle stop	Chiller unit X16
207 (F7)	IRREGULAR SLIDEWAY OIL PRESSURE	Lubrication pressure does not reduce while the grease lubrication pump is at a stop.	Stop in the single block operation mode	Pressure switch SP11 HCN 6000-II, 6800-II, 8800-II: XA5 HCN 10800-II: X65
209 (F9)	SLIDEWAY OIL MALFUNCTION	Lubrication pressure is not built up although the grease lubrication pump is operating.	Stop in the single block operation mode	Pressure switch SP11 HCN 6000-II, 6800-II, 8800-II: XA5 HCN 10800-II: X65
211 (F11)	SPINDLE CONTROLLER MALFUNCTION	Alarm related to the spindle controller	Feed hold Spindle stop	Spindle controller X323
212 (F12)	MAGAZINE DRUM MALFUNCTION	Alarm related to the magazine driver	Magazine rotation stop	Magazine driver A30 M3278, M3279 M3281, M3283 M3285
213 (F13)	INDEX TABLE DRIVER MALFUNCTION	Alarm related to the index table driver	Feed hold Spindle stop	Table driver A31 M3228, M3229 M3231, M3233 M3235
214 (F14)	ILLEGAL TOOL DESIGNATED	A tool was selected for which the tool data has been specified as invalid.	Magazine rotation stop	
217 (F17)	THERMAL TRIP	The thermal relay in one of the auxiliary motors or the 100 V circuit protector has been tripped.	Feed hold Spindle stop	KM11, KM12, KM18, KM76, QF7, QF8, X1A
218 (F18)	CONVEYOR THERMAL TRIP	The thermal relay in the motor of the chip conveyor or chip screw has been tripped.	Stop in the single block operation mode	KM15, KM16, KM13 X1B
219 (F19)	MAIN TRANSFORMER OVERHEAT	The temperature of the main power transformer has exceeded 120°C (248°F).	Stop in the single block operation mode Spindle stop	Main transformer X10
222 (F22)	AFC OVERLOAD	Spindle load does not drop even when feed override is reduced by AFC.	Feed hold	

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
223 (F23)	TOOL MEAS. UNIT NOT IN POS	An attempt has been made to perform a semi-automatic tool length measurement without the measuring unit being extended.	No operation start	
224 (F24)	ILLEGAL TOOL FOR TOOL LGTH MEAS	1. An attempt has been made to measure a touch sensor or chip removal tool. 2. An attempt has been made during a fully automatic tool length measurement to measure an inverted spot facing, boring, back boring, chamfering, face milling or special cutter.	No operation start	
225 (F25)	TOOL LIFE OVER	Accumulated application time of the tool has exceeded its specified life.	Stop in the single block operation mode if M58 is given while this alarm message is displayed. To continue operation, press the clear key and then press the cycle start button.	
226 (F26)	TOOL BREAKAGE	A broken tool has been detected by the M35 command. Alternatively, a broken tool has been mounted on the spindle.	Stop in the single block operation mode Spindle stop	
227 (F27)	SIMULTANEOUS M CODE OPERATION	M-codes have been assigned which cannot be processed simultaneously.	No operation start Machine stop	
228 (F28)	ILLEGAL M CODE	An M-code has been assigned which is not included in the specifications.	No completion signal returned, operation stop	
229 (F29)	INVALID SCREEN FOR CYCLE START	An attempt has been made to start a memory or tape operation with a display other than POSITION or TRACE on the screen.	Feed hold	
230 (F30)	ILLEGAL MMS UNIT	Some form of trouble has occurred in the MMS unit and the required signal was not output.	Feed hold Spindle stop	MMS ERROR X17 MMS LOW BATT X18
232 (F32)	MACHINE DOOR INTERLOCK	The machine door has been opened.	Feed hold Spindle stop Stop in the single block operation mode, etc.	SQ9 X07
233 (F33)	MAGAZINE DOOR INTERLOCK	The magazine door was opened in the automatic mode of operation.	Magazine rotation is disabled.	SQ31 HCN 6000-II, 6800-II: X8F HCN 8800-II, 10800-II: X28F
234 (F34)	PALLET CHANGER DOOR INTERLOCK	The pallet changer safety door has been opened, or the door lock has been released.	Pallet changer operation is disabled.	SQ56 X9C
235 (F35)	MAGAZINE MANUAL INTERRUPT	Manual interruption was made valid by the switch concerned on the magazine side.	Only display of message (Automatic magazine indexing suspended [NC call stop])	SA4 HCN 6000-II, 6800-II: X92 HCN 8800-II, 10800-II: X292
236 (F36)	HOLD PALLET CHANGE	Pallet change command was given with the machine menu displayed on the screen.	Only display of message (Automatic pallet change suspended [NC call stop])	
237 (F37)	MAGAZINE DOOR OPENED	The magazine door is open.	Only display of message	SQ31 HCN 6000-II, 6800-II: X8F HCN 8800-II, 10800-II: X28F

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
238 (F38)	AUTO TOOL CHANGE STOP	The operation has been stopped according to the selection of the [ATC STOP] menu function.	Stop just before the ATC cycle. Press the [ATC STOP] menu key to cancel the function, and the operation will be continued from the ATC cycle.	
240 (F40)	OPERATION RESTART UNFINISHED	An attempt was made to start restart operation without having completed the EIA restart setting.	Feed hold	
241 (F41)	SPINDLE ROTATION ABNORMAL	The zero speed signal remains ON even though a spindle rotation command has been given.	Feed hold Spindle stop	Y395, X324
242 (F42)	SPINDLE ORIENT TIME OVER	The spindle orient command does not result in the orient check signal being turned ON within 10 seconds.	Waiting for input of spindle orient check signal 1	Y3A6, X326
244 (F44)	ZERO SIGNAL OFF IN SPDL ORIENT	The spindle zero speed check signal turned OFF while the orient check signal was ON.	Only display of message	Y3A6, X324
246 (F46)	VERIFY TL DATA SP-SPEED SETTING	The specified spindle speed is over the safety speed calculated using the tool data.	Spindle speed restricted or Feed hold	
250 (F50)	TOOL LENGTH EXTEND SENSOR MALF.	The extension sensor does not turn ON in spite of the command for extending the measuring unit. Alternatively, the extension sensor does not turn OFF in spite of the command for retracting the measuring unit. (20 seconds)	Stop in the single block operation mode	SQ40 HCN 6000-II, 6800-II, 8800-II: XAC HCN 10800-II: X6C
251 (F51)	TOOL LENGTH RETRACT SENSOR MALF	The retraction sensor does not turn ON in spite of the command for retracting the measuring unit. Alternatively, the retraction sensor does not turn OFF in spite of the command for extending the measuring unit. (20 seconds)	Stop in the single block operation mode	SQ41 HCN 6000-II, 6800-II, 8800-II: XAD HCN 10800-II: X6D
256 (F56)	TOOL CLAMP SENSOR MALFUNCTION	The clamp sensor does not turn ON in spite of a tool clamp command. Alternatively, it does not turn OFF in spite of a tool unclamp command. (20 seconds) (5 seconds for HSK tool)	Feed hold Spindle stop	SQ11 HCN 6000-II, 6800-II, 8800-II: XA1 HCN 10800-II: X61
257 (F57)	TOOL UNCLAMP SENSOR MALFUNCTION	The unclamp sensor does not turn ON in spite of a tool unclamp command. Alternatively, it does not turn OFF in spite of a tool clamp command. (20 seconds) (5 seconds for HSK tool)	Feed hold Spindle stop	SQ10 HCN 6000-II, 6800-II, 8800-II: XA0 HCN 10800-II: X60
258 (F58)	ATC COVER SENSOR (OPEN) MALF.	The open sensor does not turn ON in spite of the command for opening the ATC cover. (30 seconds)	Stop in the single block operation mode	SQ22 HCN 6000-II, 6800-II: X88 HCN 8800-II, 10800-II: X288
259 (F59)	ATC COVER SENSOR (CLOSE) MALF.	The close sensor does not turn ON in spite of the command for closing the ATC cover. (30 seconds)	Stop in the single block operation mode	SQ23 HCN 6000-II, 6800-II: X89 HCN 8800-II, 10800-II: X289
266 (F66)	PALLET CLAMP SENSOR MALF.	The clamp sensor does not turn ON in spite of a pallet clamp command. Alternatively, it does not turn OFF in spite of a pallet unclamp command.	Waiting for the sensor input signal (Clamp detected with sensor OFF)	SP54 HCN 6000-II, 6800-II, 8800-II: XA8 HCN 10800-II: X68

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
267 (F67)	PALLET UNCLAMP SENSOR MALF.	The unclamp sensor does not turn ON in spite of a pallet unclamp command. Alternatively, it does not turn OFF in spite of a pallet clamp command	Waiting for the sensor input signal (Unclamp detected with sensor OFF)	SP55 HCN 6000-II, 6800-II, 8800-II: XA9 HCN 10800-II: X69
269 (F69)	MAGAZINE STOP PIN SENSOR MALF.	The stop pin sensor does not turn ON although magazine stop pin operation has been specified.	Waiting for sensor input signal	
270 (F70)	X AXIS ORIGIN RETURN UNFINISH	Without initial zero-point return for the X-axis, a command was given for ATC, tool length measurement, pallet change, etc.	No operation start	
271 (F71)	Y AXIS ORIGIN RETURN UNFINISH	Without initial zero-point return for the Y-axis, a command was given for ATC, tool length measurement, pallet change, etc.	No operation start	
272 (F72)	Z AXIS ORIGIN RETURN UNFINISH	Without initial zero-point return for the Z-axis, a command was given for ATC, tool length measurement, pallet change, etc.	No operation start	
273 (F73)	4 AXIS ORIGIN RETURN UNFINISH	Without initial zero-point return for the 4th axis, a command was given for ATC, tool length measurement, pallet change, etc.	No operation start	
274 (F74)	SPDL CANNOT ROT (UNCLAMPED TOOL)	A spindle rotation command was given without the tool being clamped.	Spindle rotation disabled	HCN 6000-II, 6800-II, 8800-II: SQ11, XA1 SQ10, XA0 HCN 10800-II: SQ11, X61 SQ10, X60
275 (F75)	SPDL CANNOT ROT (SPECIAL TOOL)	A spindle rotation command was given with a special tool (touch sensor, chip removing tool) on the spindle.	Spindle rotation disabled	
276 (F76)	SPDL CANNOT ROT (ATC ARM POS.)	The spindle cannot be started since the ATC arm is not positioned appropriately.	Spindle rotation disabled	
280 (F80)	NO ORIENT (UNCLAMPED TOOL)	A spindle orient command was given without the tool being clamped.	Spindle orientation disabled	HCN 6000-II, 6800-II, 8800-II: SQ11, XA1 SQ10, XA0 HCN 10800-II: SQ11, X61 SQ10, X60
281 (F81)	NO SPDL ORIENT (SPECIAL TOOL)	A spindle orient command was given with a special tool (touch sensor, chip removing tool) on the spindle.	Spindle orientation disabled	
282 (F82)	NO ORIENT (ATC ARM POS. ERROR)	A spindle orient command was given without the ATC arm being positioned appropriately.	Spindle orientation disabled	
283 (F83)	NOT UNCLAMPED (NOT STOP SPINDLE)	The tool unclamp switch was operated when the [SPDL JOG] menu key was pressed or when the spindle was rotating.	Tool unclamp disabled	M3502
284 (F84)	NOT UNCLAMPED (SPDL ORI. UNFIN)	The tool unclamp switch was operated without the spindle being oriented.	Tool unclamp disabled	Y3A6 X326
285 (F85)	NOT UNCLAMPED (NOT MANUAL MODE)	The tool unclamp switch was operated in a mode other than manual.	Tool unclamp disabled	
286 (F86)	NOT AUTO MODE (UNCLAMPED TOOL)	An attempt was made to select the automatic mode with the tool still unclamped.	Mode change disabled	HCN 6000-II, 6800-II, 8800-II: SQ11, XA1 SQ10, XA0 HCN 10800-II: SQ11, X61 SQ10, X60

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
287 (F87)	NOT AUTO MODE (MAG. MANUAL OPER)	An attempt was made to select the automatic mode during manual magazine operation.	Mode change disabled	SA4 HCN 6000-II, 6800-II: X92 HCN 8800-II, 10800-II: X292
289 (F89)	NO TOOL LOAD (ARM/SFTER POS ERR)	A tool loading command was given without the ATC arm being positioned correctly.	Operation is disabled.	HCN 6000-II, 6800-II: 40T SQ26, X80 SQ25, X82 80T SQ36, X81 SQ37, X82 HCN 8800-II, 10800-II: 40T SQ26, X280 SQ25, X282 80T SQ36, X281 SQ37, X282
290 (F90)	NO TOOL LOAD (MAGAZINE POS ERR)	A tool loading command was given without the magazine being indexed correctly.	Operation is disabled.	
291 (F91)	NO TOOL UNLOAD (TOOL IN MAG.)	A tool unloading command was given with an occupied magazine pocket being indexed.	Operation is disabled.	SQ34 HCN 6000-II, 6800-II: X84 HCN 8800-II, 10800-II: X284
292 (F92)	NO UNLOAD (ARM/SHIFTER POS ERR)	A tool unloading command was given without the ATC arm being positioned correctly.	Operation is disabled.	HCN 6000-II, 6800-II: 40T SQ26, X80 SQ25, X82 80T SQ36, X81 SQ37, X82 HCN 8800-II, 10800-II: 40T SQ26, X280 SQ25, X282 80T SQ36, X281 SQ37, X282
293 (F93)	NO TOOL UNLOAD (MAG. POS ERROR)	A tool unloading command was given without the magazine being indexed correctly.	Operation is disabled.	
294 (F94)	NO TOOL SELECT (INCORRECT TNo.)	Magazine indexing was not possible since the specified tool number was zero, or higher than the number of the magazine pockets provided.	Operation is disabled.	
295 (F95)	NO TOOL SELECT (ARM/SFT POS ERR)	A magazine indexing command is given without the shifter being positioned correctly.	Operation is disabled.	
296 (F96)	NO TOOLCHANGE (AXIS NOT ATC POS)	An EIA program's tool change command (which does not include the positioning in question) is not preceded by the required axis movement commands for the ATC position.	Operation is disabled.	

3

ALARM

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
300 (F100)	MAGAZINE SHIFTER POS.MALF.	A loading or unloading command for the magazine shifter has not been completed within 10 seconds. The magazine shifter was not stopped at the correct stand-by position.	Waiting for the sensor input signal	HCN 6000-II, 6800-II: 40T SQ26, X80 SQ25, X82 80T SQ36, X81 SQ37, X82 HCN 8800-II, 10800-II: 40T SQ26, X280 SQ25, X282 80T SQ36, X281 SQ37, X282
301 (F101)	MAGAZINE SHIFTER MISS OP.	1. An ATC command was given in the state the magazine shifter is at the ATC side without a tool in the spindle nor in the ATC standby pocket (this is unusual state and, thus, tool loading is impossible). 2. An ATC command was given in the state the magazine shifter is at the magazine side with a tool held in both the spindle and the ATC standby pocket (this is unusual state and, thus, tool unloading is impossible). 3. In manual magazine shifter operation for maintenance, the ATC standby pocket is not in a position allowing the shifter operation.	Operation is disabled.	HCN 6000-II, 6800-II: 40T SQ26, X80 SQ25, X82 80T SQ36, X81 SQ37, X82 HCN 8800-II, 10800-II: 40T SQ26, X280 SQ25, X282 80T SQ36, X281 SQ37, X282
302 (F102)	ATC WAIT POCKET PROX. MALF.	A command for rotating the ATC standby pocket has not been completed within 10 seconds. The ATC standby pocket was not stopped at the correct stand-by position.	Waiting for the sensor input signal	HCN 6000-II, 6800-II: SQ27, X85 SQ28, X86 HCN 8800-II, 10800-II: SQ27, X285 SQ28, X286
303 (F103)	ATC WAIT POCKET MISS OP.	A command for rotating the ATC standby pocket was given although the ATC arm or the magazine shifter was not at the correct position.	Operation is disabled.	HCN 6000-II, 6800-II: SQ36, X81 SQ37, X82 HCN 8800-II, 10800-II: SQ36, X281 SQ37, X282
304 (F104)	ATC ARM POSITION MALF.	The ATC arm rotation was stopped at an incorrect position. (10 seconds) The ATC operation has not been completed within 10 seconds.	Only display of message	
305 (F105)	MAGAZINE SHIFTER THERMAL TRIP	The thermal relay of the magazine shifter motor was tripped (due to an overcurrent). The alarm state cannot be eliminated until the thermal relay is reset automatically. (For magazines with 80/120 pockets only)	Operation is disabled.	FR14 HCN 6000-II, 6800-II, 8800-II: X95 HCN 10800-II: X28D
306 (F106)	ATC ARM DRIVER ALARM	An alarm occurred with the motor driver for ATC arm rotation.	Operation is disabled.	

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
307 (F107)	COVER COOLANT OFF --> ON	A warning message displayed when the memory mode is selected after switching-on the NC.	Only display of message	
308 (F108)	M.M.S. ALARM	The skip signal for the touch sensor was turned on untimely in the step of rapid traverse.	Emergency stop	Skip signal X178
309 (F109)	PALLET DRIVER ALARM	An alarm occurred with the motor driver for pallet changer rotation.	Operation is disabled.	
311 (F111)	SCALE AIR PRESSURE DOWN	The scale air pressure is low.	Only display of message	SP14 HCN 6000-II, 6800-II, 8800-II: XA3 HCN 10800-II: X63
312 (F112)	PALLET(UN)CLAMP SENSOR OFF	The upper end sensor does not turn ON during pallet arm rotation. (Only with HCN 8800-II or 10800-II)	Waiting for sensor input signal	SQ53, X90
313 (F113)	PALLET ARM DOWN RS. MALF.	The lower end sensor for the pallet arm does not come into action although the pallet arm rotation was completed.	Waiting for the sensor input signal	SQ52 HCN 6000-II, 6800-II: X9B HCN 8800-II, 10800-II: X91
314 (F114)	PALLET ARM POSITION MALF.	The pallet arm rotation was stopped at an incorrect position. (15 seconds) The pallet arm rotation has not been completed within 15 seconds. (Adjustable with HCN 10800-II only)	Only display of message	
319 (F119)	PC_MOP. (ATC COVER OPEN)	A pallet change command was given without the ATC cover being closed.	Only display of message	SQ23 HCN 6000-II, 6800-II: X89 HCN 8800-II, 10800-II: X289
320 (F120)	PC_MOP. (NOT TABLE 0 DEG)	A pallet change command was given without the table being in the 0° position.	NC call stop	
321 (F121)	PC_MOP. (M-ARM EXT.)	A pallet change command was given with the tool length measuring unit being extended.	NC call stop	HCN 6000-II, 6800-II, 8800-II: SQ40, XAC SQ41, XAD HCN 10800-II: SQ40, X6C SQ41, X6D
322 (F122)	PC_MOP. (NOT Z ZERO POSI.)	A pallet change command was given without Z-axis positioning at the second zero point.	NC call stop	
323 (F123)	PC_MOP.	Operation error for 6-pallet changer (option)	NC call stop	
		Operation error for pallet changer (FMS option)		
324 (F124)	PC_MOP. (NOT STAND 0 DEG.)	The pallet stand is not locked at the 0° position.	NC call stop	2PC SQ57, X98 FMS SQ66, X98
325 (F125)	INDEX MOP. (M-ARM EXT.)	A command for rotating the index table was given with the tool length measuring unit being extended.	NC call stop	HCN 6000-II, 6800-II, 8800-II: SQ40, XAC SQ41, XAD HCN 10800-II: SQ40, X6C SQ41, X6D
326 (F126)	INDEX MOP. (CMD TOO LARGE)	The angle specified in a command for rotating the index table was out of setting range: $0 \leq \text{CMD} \leq 359$.	NC call stop	

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
327 (F127)	INDEX MISS OP.	A command for rotating the index table was given with the pallet being unclamped. A command for rotating the index table was given although NC rotary table (optional) was used.	Only display of message	X23C
328 (F128)	PALLET 1 CYCLE STOP	A table index command was given although the pallet was unclamped.	Machining cycle can be started by turning on the 1 cycle lamp by pressing the 1 cycle switch after mounting workpieces on the pallet.	
329 (F129)	TOOL CHANGE MISS OP.	A tool change command was given although the ATC arm was not in the correct position.	Arm is not at 0° position. Arm is not retracted.	
		A manual tool change command was given although the magazine shifter was not at the waiting position.	Shifter is not at waiting position.	SQ37 HCN 6000-II, 6800-II, 8800-II: X82 HCN 10800-II: X282
330 (F130)	PALLET CLAMP/ UNCLAMP MISS OP.	Pallet clamp or unclamp was attempted although the pallet changer hook was not at the required position.	Pallet change hook not at in/out end.	
		An attempt was made to unclamp a pallet without positioning on the Z-axis or the 4th-axis (or for the index table) to the pallet change position nor the table being clamped.	No Z-axis positioning to the second zero point. Table not at 0°.	
			Table not clamped.	
331 (F131)	MULTI PALLET MISS OP.	Multi pallet change operation error. See Operating Manual for Multiple Pallet Changer for details.	Only display of message	
332 (F132)	MULTI PALLET ALARM	Multi pallet change alarm See Operating Manual for Multiple Pallet Changer for details.	Only display of message	
333 (F133)	SET UP SW.MISS.OP. (SP.ORIENT)	An attempt was made to turn the machine set up switch to the "I" position with the spindle being stopped in the orientation position.	The mode is not changed to the set up mode.	
334 (F134)	ID AMP COMMUNICATION TIME OVER	Communications with the ID controller have been off for 60 seconds.	Only display of message	
335 (F135)	TABLE UNCLAMP MISS OP.	The NC table rotation command was given although the pallet was not clamped.	Only display of message	
336 (F136)	ANALOG INPUT ERR. (LEVEL SENSOR)	Analog input voltage too high. (Alarm for 0 V line disconnection)	Only display of message	R2092 bit 5
		Analog input voltage too low. (Alarm for +24 V signal line disconnection) (option)	Only display of message	R2092 bit 6
337 (F137)	CURRENT TIME IS NOT SET.	Current time is not set. (expanded self-diagnose function; option)	Only display of message	R2092 bit F
338 (F138)	PLEASE EXCHANGE OIL(HYD. UNIT)	Oil in the hydraulic unit must be changed. (Expanded self-diagnose function; option)	Only display of message	R2092 bit 0
339 (F139)	PLEASE EXCHANGE OIL(SP. LUBE)	Spindle lubricating oil must be changed. (Expanded self-diagnose function; option)	Only display of message	R2092 bit 1
340 (F140)	PLEASE SUPPLY OIL (WAY LUB-UNIT)	Slideway lubricating oil level is low. (option)	Only display of message	R2283
341 (F141)	PLEASE SUPPLY OIL (COOLANT TANK)	Coolant level is low. (Expanded self-diagnose function; option)	Only display of message	R2092 bit B
342 (F142)	PLEASE SUPPLY OIL (SP.LUBE)	Spindle lubricating oil level is low. (Expanded self-diagnose function; option)	Only display of message	R2092 bit 9 or C
344 (F144)	PARAMETER ERROR RS51-RS60	A pocket number that cannot accommodate a large diameter tool is set for parameters RS51 to RS60.	Only display of message	

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
345 (F145)	CAN'T CLOSE ATC COVER_BIG TOOL	With the ATC cover open and a large diameter tool mounted in the spindle, an ATC cover close command was specified while the X-axis was in the range where interference between the ATC cover and the large diameter tool could occur.	Only display of message	
348 (F148)	PALLET UNCLAMP MISS OP.	Pallet unclamp command was given with the table being unclamped.	Pallet unclamp suspended.	SQ44 (Index table) SP2 (NC rotary table) HCN 6000-II, 6800-II, 8800-II: XAA HCN 10800-II: X6A
349 (F149)	TRANSMIT PUMP OFF->ON	High-pressure coolant (option)	Stop in the single block operation mode	
351 (F151)	TOUCH SENSOR OFF	The touch sensor does not turn ON. (option)	Only display of message	
354 (F154)	CHIP CONVEYOR ALARM	The shock relay detected over-current of the chip conveyor motor (option).	Single block stop	Chip conveyor X0C
355 (F155)	OIL AIR UNIT PRESSURE DOWN	The hydraulic pressure signal of the oil-air unit is not turned ON/OFF.	Feed hold Spindle stop	SP3 HCN 6000-II, 6800-II, 8800-II: XAF HCN 10800-II: X6F
356 (F156)	OIL AIR UNIT LEVEL DOWN	Oil-air unit lubricating oil level is too low.	Only display of message	SL3 HCN 6000-II, 6800-II, 8800-II: XAE HCN 10800-II: X6E
357 (F157)	OIL AIR UNIT ALARM	Oil-air unit lubricating oil level is left too low.	Feed hold Spindle stop	SL3 HCN 6000-II, 6800-II, 8800-II: XAE HCN 10800-II: X6E
358 (F158)	LINE FILTER MALFUNCTION	Filter clogged signal does not remain ON for 5 seconds or longer. (option)	Only display of message	SP7, SP8, X0
359 (F159)	LINE FILTER ALARM	After the occurrence of F158, signal OFF state has continued for more than 3 hours. (option)	Stop in the single block operation mode Spindle stop	SP7, SP8, X0
362 (F162)	HIGH PRE COOL SYS.THERMAL TRIP	A thermal trip alarm has occurred in the high-pressure through-spindle coolant unit. (option)	Feed hold Spindle stop	
363 (F163)	HIGH PRE COOL SYS.PRE ALARM	A pressure alarm has occurred in the high-pressure through-spindle coolant unit. (option)	Feed hold Spindle stop	
365 (F165)	TOOL HIVE DOOR INTERLOCK	The cycle start button was pressed while the TOOL HIVE door was open. (option)	Only display of message	
366 (F166)	TOOL HIVE CONTROLLER ALARM	Alarm related to TOOL HIVE (option)	TOOL HIVE operation is disabled.	
367 (F167)	TOOL HIVE MOVING	Manual ATC operation was attempted while the TOOL HIVE was operating (option).	Only display of message	
368 (F168)	TOOL HIVE INTERRUPT	Manual TOOL HIVE interruption (option)	Only display of message	

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
369 (F169)	WARNING SET UP SW. ON!	An attempt was made in the automatic operation mode to turn the machine set up switch to the "I" position.	Only display of message	SA6, X11C
370 (F170)	SPINDLE TRIAL RUN !!	Trial run is required (high-speed spindle; option).	Only display of message	
372 (F172)	HIGH PRESSURE UNIT ALARM	Alarm occurred with the coolant unit (option).	Feed hold Spindle stop	
373 (F173)	WAITING TOOL NUMBER ILLEGAL	In the automatic mode, 0 is displayed as the pocket number with the tool pot at the ATC side.	Only display of message	
374 (F174)	COOLANT LEVEL TOO LOW	Coolant level is low (option).	Feed hold Spindle stop	X0E
375 (F175)	COOLANT LEVEL LOW WARNING	Although the suction pump is turned on, coolant tank level is not raised (option).	Only display of message	
376 (F176)	WORK NO. SEARCH ERROR	The program specified for warm-up operation is not found.	Only display of message	R487
377 (F177)	WORK NO.SET ERROR	The program number of the program for warm-up operation is illegal: 0 or exceeding 9999.	NC call stop	R488 RS5 (R10984)
378 (F178)	WARMING UP !!	(High-speed spindle option)	Only display of message	
379 (F179)	SPINDLE OVER LOAD	(High-speed spindle option)	Only display of message	
380 (F180)	PUSH MF1 TO READ SPINDLE LOAD		Only display of message	
381 (F181)	IMPOS. T-ID READ (T-DATA)	Tool ID readout was attempted though tool data exists. (R3728 = 0FF2H)	Only display of message	R3728
382 (F182)	IMPOS. T-ID READ (NO TOOL)	Tool ID readout was attempted though no tool or a tool without tool ID was mounted. The tool ID head not at the readout position. (R3728 = 0FF5H)	Only display of message	R3728
383 (F183)	TOOL ID READ ALARM	Alarm No. 381 occurred. (R3728 = 0FF2H) Alarm No. 382 occurred. (R3728 = 0FF5H) Tool ID data readout not completed within the preset time. (R3728 = 0FF1H) Response signal remaining on. (R3728 = 0FF4H) Communication impossible. (Response signal remaining off). (R3728 = 0FF6H) Tool data exists. Tool ID exists. (Readout according to the command from tool transport system failed.)	Only display of message	R3728
384 (F184)	IMPOS. T-ID WRITE (NO DATA)	Tool ID data writing was attempted without tool data. (R3792 = 0H)	Only display of message	R3792
385 (F185)	IMPOS. T-ID WRITE (NO TOOL)	Tool ID data writing was attempted without a tool or a tool not equipped with tool ID. The tool ID head not at the writing position. (R2554 = 0D6CH)	Only display of message	

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
386 (F186)	TOOL ID WRITE ALARM	Alarm No. 384 occurred. (R2080 = 201H) Alarm No. 385 occurred. (R2080 = 202H) Tool ID data writing not completed within the preset time. (R2080 = 220H) Response signal remaining on. (R2080 = 221H) Read tool No. is 0 or tool large. (R2080 = 222H) Communication impossible. (Response signal remaining off). (R2080 = 223H) No tool data. (Writing according to the command from tool transport system failed.) (R2080 = 210H)	Only display of message	R2080
388 (F188)	PALLET ID ALARM		Only display of message	R2018
389 (F189)	PALLET ID UP/DOWN SENSOR MALF.	Pallet ID head upper or lower end detection sensor does not go ON.	Only display of message	
390 (F190)	U-AXIS ORIGIN RETURN MISS OP.	The specified U-axis zero-point return direction is illegal.	Only display of message	
391 (F191)	THERMAL DISPLACEMENT SYSTEM ER	Alarm No. 392 occurred 10 times.	Feed hold Spindle stop	R4630 to R4639
392 (F192)	THERMAL DISPLACEMENT OFFSET ER	Spindle over heat alarm Spindle temperature raised by 2°C (3.6°F) or more in approx. 6 min. (R2033 bit 0: ON) Spindle temperature alarm Temperature difference between spindle and base became 20°C (36°F) or more. (R2033 bit 1: ON) Parameter error Thermal displacement compensation parameter out of specified range. (R2033 bit 2: ON) Hardware error Disconnection, short circuit etc. occurred in the base side temperature sensor. (R2033 bit 3: ON) Breakage or short circuit occurred in the spindle side temperature sensor. (R2033 bit 4: ON)	Feed hold Spindle stop The interruption of operation by this alarm can be suppressed by setting RB10 bit 3 to "1".	R2033
394 (F194)	COOLANT STOP!!	The cycle start button is pressed when the coolant stop button is ON.	Only display of message	
395 (F195)	TOUCH SENSOR BATTERY LEVEL LOW	The touch sensor battery voltage has dropped.	Feed hold Spindle stop	
396 (F196)	SAVING ENERGY MODE	The machine entered the standby mode.	Only display of message	
397 (F197)	TABLE CLAMP PRS. ALARM	The table clamp sensor is not turned on although the table clamp command is assigned, or the table clamp sensor is not turned off although the table unclamp command is assigned.	Waiting for the sensor input signal	SQ44 (Index table) SP2 (NC rotary table) HCN 6000-II, 6800-II, 8800-II: XAA HCN 10800-II, X6A

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
398 (F198)	OPERATION SWITCH MALFUNCTION	The status of any of the switches below is abnormal before the power is turned on. <ul style="list-style-type: none"> • Cycle start button • Spindle start button • Spindle JOG switch • Magazine FORWARD/REVERSE button • Tool unclamp switch • Machine set up switch • Axis movement buttons • Pallet lock pin switch (UNLOCK) 	Emergency stop	
399 (F199)	POWER CUT OFF MALFUNCTION	Power shut-off contactor malfunction Improper adjustment of the door limit switch The door is not fully closed.	Emergency stop	KM1, KM2, KM3, KM4, KM9, KM8, KM7, KM17 SQ9, SQ31, SQ56
1210 (F210)	TABLE CLEANUP MIS-OPERATION	The [TABLE CLEANUP MODE]/[TABLE CLEANUP RELEASE] menu key is pressed while the relevant conditions are not satisfied.	Only display of message	
1213 (F213)	MAGAZINE POT POS.MALF.	The locking/unlocking operation of the ATC pot does not complete.	Only display of message	
1214 (F214)	TABLE POSITION ALARM	The table has been displaced. (Only when the NC positioning table is equipped) Contact the nearest Mazak Technical Center or Technology Center.	Feed hold Spindle stop	
1217 (F217)	SUPPLY LUBLICANT TO CLAMP CORN	The number of tool unclamping operations has exceeded 36000. (option)	Only display of message	
1219 (F219)	COOLANT TEMP CONT. FREEZE ALARM	The coolant temperature has dropped below 3°C (37°F). (option)	Only display of message	HCN 6000-II, 6800-II, 8800-II: X26 HCN 10800-II: X25
1220 (F220)	PC_MOP.(PALLET ARM CW/CCW)	The [PC ARM CCW] menu key has been pressed with pallet No. 1 located inside the machine or the [PC ARM CW] menu key has been pressed with pallet No. 2 located inside the machine.	Pallet change is disabled.	
1221 (F221)	PC_MOP.(PC POS. MALF.)	The pallet arm position has exceeded the software limit. (Only with HCN 8800-II or 10800-II)	Pallet change is disabled.	
1222 (F222)	PC_MOP.(PALLET UNCLAMP)	A pallet change command was specified with the pallet unclamped.	Only display of message	
1224 (F224)	TAP RETRACT NOT POSSIBLE	The [TAP RETRACT] menu key was pressed although the conditions for retracting the tap were not satisfied. (option)	Feed hold Spindle stop	X246

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
1225 (F225)	POCKET NUMBER REPETITION	<p>This alarm occurs when an abnormality (duplication) is found in the pocket numbers before or after a tool change operation. (The pocket number specified for an overwriting request is duplicated.)</p> <ol style="list-style-type: none"> If two or more instances of the same pocket number are found before a tool change, "4" is set for R2036. If one or more of the same pocket number is found after a tool change, "4" is set for R2040. <p>Take note of the above data and contact the nearest Mazak Technical Center or Technology Center.</p> <p>If the tool data differ from the actual tool positions, match the data with the actual status and start the automatic operation.</p>	Feed hold Spindle stop	
1226 (F226)	TOOL NUMBER INVALIDITY	<p>This alarm occurs when an invalid tool number is found before or after a tool change operation. (The specified tool number is not registered in the tool data.)</p> <ol style="list-style-type: none"> If an invalid tool number is found before a tool change, "5" is set for R2036. If an invalid tool number is found after a tool change, "5" is set for R2040. <p>Take note of the above data and contact the nearest Mazak Technical Center or Technology Center.</p> <p>If the tool data differ from the actual tool positions, match the data with the actual status and start the automatic operation.</p>	Feed hold Spindle stop	
1227 (F227)	DESIGNATED POCKET No.INCORRECT	<p>This alarm occurs when the overwritten pocket numbers coincide before and after a tool change.</p> <ul style="list-style-type: none"> Overwritten pocket number before tool change: R2038 Overwritten pocket number after tool change: R2041 <p>Take note of the above data and contact the nearest Mazak Technical Center or Technology Center.</p> <p>If the tool data differ from the actual tool positions, match the data with the actual status and start the automatic operation.</p>	Feed hold Spindle stop	
1229 (F229)	CHILLER UNIT COOLING MALF.	The spindle temperature has risen too much.	NC call stop	
1231 (F231)	FILTER ALARM	The filter clogged signal does not turn ON for 5 seconds or longer. (option)	Stop in the single block operation mode Spindle stop	SP37, X0
1240 (F240)	ATC RECOVERY MODE IMPOSSIBLE	The [ATC RECOVERY MODE] menu key is pressed while automatic recovery is not possible after the ATC operation has stopped partway. Recover manually.	Only display of message	
1241 (F241)	PC RECOVERY MODE IMPOSSIBLE	The [PC RECOVERY MODE] menu key is pressed while automatic recovery is not possible after the APC operation has stopped partway. Recover manually.	Only display of message	
1250 (F250)	PC_STAND DRIVER ALARM	An alarm occurred with the pallet rotation stand motor driver. (Only with HCN 10800-II)	Operation is disabled.	A36

Alarm No. (SQ coil No.)	Alarm message	Description	Machine status	Alarm related detector SQ No.
1251 (F251)	OPEN THE ATC COVER	The ATC cover needs to be opened once using the manual operation menu. (Only with HCN 10800-II)	Operation is disabled.	SQ22, X288 SQ23, X289
1255 (F255)	MRJ2 BATTERY LEVEL LOW	The battery for the auxiliary motor is exhausted.	Only display of message	A30, A32, A33, A36
1256 (F256)	NOT CYCLE START. (LINE FILTER ALARM)	2 hours and 30 minutes has elapsed after turning off the filter clogging signal. (Only with HCN 10800-II)	NC call stop	SP7, SP8, X0
1257 (F257)	TOOL HIVE DOOR OPEN	The TOOL HIVE door is open.	TOOL HIVE operation is disabled.	
1258 (F258)	TOOL HIVE WAIT TIME	Waiting for a response from TOOL HIVE	Waiting for input signal	
1271 (F271)	OVER LOAD	A load beyond the preset value has been detected on the X/Y/-Z-/B-axis or the spindle. (option)	Feed hold Spindle stop	
1272 (F272)	OVER LOAD DETECT MISS OP.	A value larger than the threshold is set for overload detection. (option)	NC call stop	
1277 (F277)	TOOL No.OK? (SPDL,NEXT,) ----->	This alarm message is displayed when automatic ATC recovery is finished. The detail code indicates (from the left): the spindle tool number and the standby tool number.	ATC operation prohibited Cycle start prohibited	
1278 (F278)	ATC RECOVERY MODE SELECTED	This alarm message is displayed when ATC recovery mode is selected.	All axes interlock Mode change is disabled.	
1280 (F280)	PALLET No.OK? (IN, ,) ----->	This alarm message is displayed when automatic PC recovery is finished. The detail code indicates: the pallet number in the machine.	PC operation prohibited Cycle start prohibited	
1281 (F281)	PC RECOVERY MODE SELECTED	This alarm message is displayed when PC recovery mode is selected.	All axes interlock Mode change is disabled.	
1288 (F288)	MISS OP.(TOOL NOT MOUNTING)	The M166 or M167 command was given when there was no tool in the spindle. (option)	Only display of message	
1289 (F289)	MISS OP.(RUNOUT DET.MNL.MODE)	The M166 or M167 command was given when the runout detection unit was not in the automatic mode. (option)	Only display of message	
1290 (F290)	MISS OP.(SP.OVERRIDE ZERO)	The M166 or M167 command was given when spindle override was set to 0 at the start of measurement. (option)	Only display of message	
1291 (F291)	SPINDLE TOOL MOUNTING FAILURE	An alarm occurs with the runout detection unit. (option)	Feed hold Spindle stop	
1300 (F300)	PLEASE CHECK THE TOOL DATA.	This message is displayed when using the tool of which data has been changed for the first time. (When the changed data check function is enabled)	Only display of message	
1301 (F301)	PLEASE CHECK THE TOOL OFFSET.	This message is displayed when using the tool of which offset has been changed for the first time. (When the changed data check function is enabled)	Only display of message	
1302 (F302)	RAPID OVERRIDE LIMIT	The rapid override value is clamped because tool data or tool offset value has been changed.	Only display of message	
1303 (F303)	ILLEGAL M-CODE	An M code unavailable due to specifications was specified.	Only display of message	
1309 (F309)	SELECT MODE3	This message is displayed when the mode 3 switch is ON. (option)	Only display of message	
1312 (F312)	SPINDLE ROTATION EXCEEDED	Spindle speed has exceeded the mode 3 allowable speed.	Feed hold Spindle stop	

3-2-5 No. 400 - No. 499, No. 1400 - No. 1499 (CNC screen operation error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display
400	— (, ,)				
Cause	—				
Action	—				
401	ILLEGAL FORMAT (, ,)	A	L	S	Blue
Cause	The format of the input data is not an available one. Example : Negative data has been input to an item that rejects negative data input.				
Action	Press the data cancellation key and then input correct data.				
402	ILLEGAL NUMBER INPUT (, ,)	A	L	S	Blue
Cause	1. The work number of a display inhibiting program was specified. 2. The numeric value that has been input is out of the allowable range.				
Action	1. The operation concerned cannot be performed for the program of display inhibition (Program management function). 2. Press the clear key and then input correct data.				
403	PROGRAM TOO LARGE (, ,)	A	L	S	Blue
Cause	The limit of 2000 lines per program has been exceeded.				
Action	Recreate the program so that it consists of 2000 lines or less.				
404	MEMORY CAPACITY EXCEEDED (, ,)	A	L	S	Blue
Cause	1. Additional creation of a machining program is no longer possible since the memory has already been filled up to its machining-program data storage capacity. 2. Additional preparation of process control data is no longer possible since 100 sets of such data have already been stored. 3. Additional preparation of program layout data is no longer possible since 4000 sets of such data have already been stored.				
Action	Make an available storage area by either erasing an unnecessary machining program from the memory or saving a machining program onto an external storage, and then create a new machining program.				
405	PROGRAM No. NOT FOUND (, ,)	A	L	S	Blue
Cause	An attempt has been made to select a program whose work number has not been registered.				
Action	Select a program whose work number has been registered.				
406	MEMORY PROTECT (, ,)	A	L	S	Blue
Cause	1. Inhibiting operation (editing, erasing, renumber of work number and entry of names) has been performed for the edit-inhibiting program. 2. PROGRAM LOCK/ENABLE switch on the operating panel is set to the LOCK position. 3. An attempt has been made to carry out "TOOL NAME ORDER" operation on the TOOL DATA display while a tool remains set in the spindle.				
Action	1. The operation concerned cannot be performed for the edit-inhibiting program (program management function). 2. Set the PROGRAM LOCK/ENABLE switch to the ENABLE position. 3. Remove the tool from the spindle, and then carry out the operation once again.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
407	DESIGNATED DATA NOT FOUND (, ,)	A	L	S	Blue
Cause	The number or character string that has been designated does not exist in the program.				
Action	Designate an existent number or character string.				
408	PROGRAM ERROR (, ,)	A	L	S	Blue
Cause	The memory contents in the machining-program data storage area have been destroyed.				
Action	Delete the corresponding program.				
409	ILLEGAL INSERTION (, ,)	A	L	S	Blue
Cause	Program data insertion is not possible.				
Action	It is not possible to insert data before the common unit.				
410	ILLEGAL DELETION (, ,)	A	L	S	Blue
Cause	Program deletion is not possible. <ul style="list-style-type: none"> • An attempt has been made to erase the common unit during editing of the MAZATROL program. 				
Action	<ul style="list-style-type: none"> It is not possible to delete the common unit. • Edit the program only after moving the cursor to the position where the data exists. 				
411	POWER OFF DURING PROGRAM EDIT (, ,)	A	L	S	Blue
Cause	A portion of the program may have been destroyed because power has been turned off during program editing.				
Action	Check the corresponding program for incorrect data, and correct the program data if an error(s) exists in it.				
412	SUB PROGRAM NESTING EXCEEDED (, ,)	A	L	S	Blue
Cause	The number of repeats of subprogram nesting has exceeded nine times.				
Action	Correct the program so that the total number of repeats of subprogram nesting becomes nine or less.				
413	MAX. NO. OF REGIST PROG EXCEEDED (, ,)	A	L	S	Blue
Cause	The program registration has exceeded its maximum value available (Standard: 256 programs).				
Action	Reduce the total number of registered programs by deleting unnecessary programs or moving them to the backup area or by saving unnecessary programs in an external storage unit and then deleting them.				
414	AUTO CALCULATION IMPOSSIBLE (Note, ,)	A	L	S	Blue
Cause	<p>Automatic calculation of surface speed and feedrate is not possible.</p> <p>Note :</p> <p>The sub-error codes displayed when the [NAVIGATE] menu function is selected on the MACHINING NAVIGATION-PREDICTION display are listed below.</p> <ul style="list-style-type: none"> -1: MAZATROL program file-opening error -2: MAZATROL program file-reading error -3: Tool materials mismatch error (when tool materials numbers are acquired) -4: Surface speed auto-setting error -5: File-opening error relating to the basic coefficients of the workpiece materials upper-limit values -6: Workpiece materials mismatch error -7: File-opening error relating to surface speed data tables -8: Tool materials mismatch error -9: Navigation file missing 				
Action	Check and correct the tool sequence data or machining unit of the program.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
415	MIS-SET G CODE (, ,)	B	L	S	Blue
Cause	A G-code not covered by the specifications has been designated.				
Action	Check and correct the G-code addresses within the program.				
416	AUTO PROCESS IMPOSSIBLE (, ,)	A	L	S	Blue
Cause	Tools cannot be automatically developed because of errors of the machining-unit data.				
Action	Check and correct the machining-unit data.				
417	EDITING PROHIBITED (, ,)	A	L	S	Blue
Cause	An attempt has been made to modify a program whose editing is prohibited.				
Action	Modify the data only after canceling the parameter setting of prohibition of editing.				
418	EIA/ISO CONVERTING (, ,)	A	L	S	Blue
Cause	During EIA/ISO conversion, an attempt has been made to perform erasure, work number change or editing of the conversion source program. Or an attempt has been made to select the TOOL PATH CHECK display.				
Action	During EIA/ISO conversion, erasure, work number change or editing of the conversion source program cannot be done. The TOOL PATH CHECK display cannot be selected.				
419	AUTO TAP PROCESS IMPOSSIBLE (, ,)	A	L	S	Blue
Cause	<ol style="list-style-type: none"> The pitch or other data cannot be automatically set because of incorrectness of the tap nominal diameter in the tapping-unit data. Although the text file is referred to for the pipe tap auto-setting (D95 bit 0 = 1), the auto-setting function cannot be executed since the auto-setting text file (Pipescdt.txt) is incorrect or contains no data. 				
Action	<ol style="list-style-type: none"> Check and correct the tapping-unit data and tapping-tool sequence data of the program. Check and correct the auto-setting text file (Pipescdt.txt). 				
420	SAME DATA EXISTS (, ,)	A	L	S	Blue
Cause	<p>An attempt has been made to input the same data as that which has already been registered.</p> <ol style="list-style-type: none"> Pocket number in the TOOL LAYOUT display. Machining-program number (changed) Machining priority number Tool model name 				
Action	<p>Check and correct the data settings.</p> <p>In case 4, this alarm indicates that the tool model name the operator has attempted registering is already present under the same tool type (such as: drill, end mill, turning tool, or grooving tool) in other systems. Specify another tool model name.</p>				
421	DATA NOT FOUND (, ,)	A	L	S	Blue
Cause	An attempt has been made to designate the data that does not exist.				
Action	Check whether the designated data exists.				
422	MEMORY PROTECT (I/O BUSY) (, ,)	A	L	S	Blue
Cause	An attempt has been made to edit or input the machining program, tool data, etc. during I/O operation.				
Action	Wait until the I/O operation is completed, and then repeat the editing or input operation from the beginning.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
423	MAX NUMBER OF TOOLS EXCEEDED (, ,)	A	L	S	Blue
Cause	During tool layout, the number of tools used in the designated program has exceeded the maximum available number.				
Action	Check and correct the corresponding machining program so that the maximum available number of tools is not exceeded.				
424	ALL POCKET NUMBERS NOT ASSIGNED (, ,)	A	L	S	Blue
Cause	It is not possible to finish the tool layout operation because the pocket number has not yet been assigned to all the required tools.				
Action	Assign the pocket number(s) and then finish the tool layout operation.				
425	DATA MISSING (, ,)	A	L	S	Blue
Cause	Processing is not possible because of lack of data. 1. Saving or loading has been attempted without designating any data (such as work numbers, etc.) on the DATA I/O display. 2. The data to be input for restart operation is wanting.				
Action	Input data correctly.				
426	PROGRAM DATA MISSING (, ,)	A	L	S	Blue
Cause	The tool sequence data cannot be automatically developed because of partial lack of the machining-unit data.				
Action	Fill up all the machining-unit data items with data.				
427	MEMORY PROTECT (AUTO MODE) (, ,)	A	L	S	Blue
Cause	An attempt has been made to input unallowable data in the automatic operation mode.				
Action	Change the mode over to the manual operation mode, and then input the data.				
428	MEMORY PROTECT (AUTO OPERATION) (, ,)	A	L	S	Blue
Cause	An attempt has been made to input unallowable data on a display (such as the TOOL DATA display) during automatic operation.				
Action	Input allowable data only after placing the NC unit in its reset state or after changing the current mode over to another mode.				
429	CANNOT PERFORM MEASUREMENT (, ,)	A	L	S	Blue
Cause	The following conditions were not satisfied: Coordinate measurement 1. Automatic operation must not be in progress. 2. The spindle must have a tool mounted on it. 3. The tool data of the tool mounted on the spindle must have already been input. Tool-length measurement 1. Automatic operation must not be in progress.				
Action	Set the specified conditions and then make the measurement.				
430	ILLEGAL TOOL DESIGNATED (, ,)	A	L	S	Blue
Cause	During creation of a machining program, an attempt has been made to input a tool name not available for the particular program unit, in the tool sequence.				
Action	Designate a correct tool name.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
431	ILLEGAL PALLET No. (, ,)	A	L	S	Blue
Cause	A nonexistent pallet number has been designated.				
Action	Designate a correct pallet number.				
432	ILLEGAL TOOL No. (, ,)	A	L	S	Blue
Cause	A nonexistent tool number has been designated.				
Action	Designate a correct tool number.				
433	SAME PROGRAM EXISTS (, ,)	A	L	S	Blue
Cause	The number of the machining program that has been designated for program reading from an external unit already exists within the NC memory.				
Action	Check the number of the machining program.				
434	NO ASSIGNED TOOL IN TOOL FILE (, ,)	A	L	S	Blue
Cause	The milling tools (face-mills, end-mills, chamfering cutters, and ball end-mills) that have been designated on the machining program include a one(s) that is not yet registered in the TOOL FILE display.				
Action	Register the corresponding tools in the TOOL FILE display.				
435	PROGRAM CHECK NOT ALLOWED (, ,)				
Cause	An attempt has been made to restart on the TOOL PATH display during checking of the tool path.				
Action	Interrupt the tool path checking operation before restarting.				
436	UNREGISTERED TNo. (, ,)	A	L	S	Blue
Cause	An unmeasurable tool number has been designated in the automatic tool-length (diameter) measurement mode.				
Action	Designate a measurable tool number.				
437	NO NOM-Ø DATA IN PROGRAM (, ,)	A	L	S	Blue
Cause	It has been found during tool layout that there is a tool without a nominal diameter in the designated program.				
Action	Check if nominal diameters have been assigned to all tools registered in the designated program.				
438	END UNIT NOT FOUND (, ,)	A	L	S	Blue
Cause	The end unit is not included in the machining program.				
Action	Create the end unit at the end of the program.				
439	MAZATROL PROGRAM DESIGNATED (, ,)	A	L	S	Blue
Cause	A MAZATROL program has been designated for copying purposes during EIA/ISO program editing.				
Action	No MAZATROL programs can be designated for copying purposes during EIA/ISO program editing.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
440	EIA/ISO PROGRAM DESIGNATED (, ,)	A	L	S	Blue
Cause	1. The machining program that has been designated on the TOOL LAYOUT, PROCESS CONTROL or SET UP RECORD display is an EIA/ISO program. 2. An EIA/ISO program has been designated for copying purposes during MAZATROL program editing. 3. An EIA/ISO program has been designated as the source program of EIA/ISO conversion. 4. An EIA/ISO program has been designated when writing coordinate values on the MEASURE display.				
Action	No EIA/ISO programs can be designated for operation on the TOOL LAYOUT, PROCESS CONTROL, SET UP RECORD or MEASURE display or during EIA/ISO conversion or MAZATROL program editing.				
441	UNREGISTERED HEAD DATA (, ,)	B	L	S	Blue
Cause	The head number that has been designated during MDI-MMS setting does not exist in the head data.				
Action	Review the designated head number.				
442	DATA RENEWAL NOT ALLOWED (, ,)	A	L	S	Blue
Cause	No updates can be made to the machining program.				
Action	This message may also be displayed when the NC unit is busy processing data. Press the clear key and then carry out the operation once again.				
443	HELP IS NOT AVAILABLE (, ,)	A	L	S	Blue
Cause	No help display is prepared for the line on which the cursor is placed.				
Action	Refer to the Programming Manual (MAZATROL Program).				
444	EDITING PROHIBITED AREA (, ,)	A	L	S	Blue
Cause	During automatic operation based on the EIA MONITOR display, an attempt has been made to move the cursor to the program section whose editing was prohibited.				
Action	The cursor cannot be moved to the area where editing is prohibited.				
445	ILLEGAL UNIT (, ,)	B	L	S	Blue
Cause	An attempt has been made to set tool layout data in a MAZATROL program containing an illegal unit(s).				
Action	Review the program.				
446	RESTART TIMES EXCEEDED (, ,)	A	L	S	Blue
Cause	The block to be searched for at the time of restart of the EIA/ISO program does exist, but the designated number of times of reappearance of the block is too large.				
Action	Check the number of times of reappearance of the block.				
447	PROGRAM ERROR (, ,)	A	L	S	Blue
Cause	A program error(s) has occurred during EIA/ISO restart search.				
Action	The program being searched for includes an error(s). Perform a tool-path check upon the program contents.				
448	RESTART SEARCH UNFINISHED (, ,)				
Cause	EIA/ISO restart searching has not been executed.				
Action	Designate the restart position and press the [EIA/ISO SEARCH] button to search the intended restart position.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
449	RESTART SEARCH FINISHED (, ,)	A	L	S	Blue
Cause	An attempt has been made to carry out another search operation when EIA/ISO restart searching had already been finished. An attempt has been made to change the work number when EIA/ISO restart searching had already been finished.				
Action	Press the reset key and then carry out the restart operation once again. Before changing the work number, press the reset key.				
450	TOUCH SENSOR NOT IN SPINDLE (, ,)	A	L	S	Blue
Cause	The spindle did not have a mounted touch sensor when an attempt was made to set MAZATROL coordinate measurement data on the PROGRAM (MAZATROL) display.				
Action	Mount a touch sensor in the spindle before setting the data.				
451	SAME MATERIAL ENTERED (, ,)	B	L	S	Blue
Cause	The materials name that has been designated on the CUTTING CONDITION - W.-MAT/T.-MAT. display already exists.				
Action	Designate a new materials name.				
452	NO SHAPE DATA IN UNIT (, ,)	A	L	S	Blue
Cause	No shape data exists in the program unit that has been designated in an attempt to make a copy of shape data.				
Action	Check the contents of the program unit for which shape copying is to be made.				
453	NO SHAPE DATA TO COPY IN UNIT (, ,)	A	L	S	Blue
Cause	An attempt has been made to copy shape data whose type is not available for the particular program unit.				
Action	It is not possible to copy shape data of the pallet-changing unit, index unit, or other units that do not have a shape sequence.				
454	CURSOR POSITION INCORRECT (, ,)	A	L	S	Blue
Cause	Processing not permissible for the current cursor position has been attempted. Example : 1. An attempt has been made to carry out a shape copying operation with the cursor on the tool sequence line. 2. The SHAPE CHECK display has been selected on a shape sequence line not actually executed during automatic operation.				
Action	Example : 1. No shape data can be copied on the tool sequence line. 2. Review the program.				
455	SAME PROGRAM No. DESIGNATED (, ,)	A	L	S	Blue
Cause	The machining program currently being edited has been appointed for the particular program copying operation.				
Action	Copying within the same program is not possible. Check the designated program number.				
456	NO TOOL IN SPINDLE (, ,)	A	L	S	Blue
Cause	The spindle does not currently have a tool mounted on it.				
Action	After mounting a tool on the spindle, carry out the particular operation once again.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
457	DATA ADDRESS NOT FOUND (, ,)	A	L	S	Blue
Cause	During creation of manual program mode unit, data setting has been attempted without addressing.				
Action	During creation of the manual program mode unit, designate an address before setting data.				
458	INTERFERING TOOL REGISTERED (, ,)				
Cause	An attempt has been made to register a tool most likely to interfere with an adjacent pocket. Example : 1. An attempt has been made to register tool data or tool change data on the TOOL DATA display. 2. An attempt has been made to set such a tool on the TOOL LAYOUT display that is likely to interfere with an adjacent pocket.				
Action	Select a pocket that does not cause interference with an adjacent one.				
459	DISPLAY PROTECT (, ,)	A	L	S	Blue
Cause	An attempt has been made to display a program whose display is prohibited.				
Action	Display the program only after canceling the parameter setting of prohibition of display.				
460	— (, ,)				
Cause	—				
Action	—				
461	PRIORITY No. OVERLAP (WNo., UNo., SNo.)	A	L	S	Blue
Cause	The same priority number is assigned to different tools.				
Action	Within one process, the same priority number must not be assigned to different tools. Change the priority number.				
462	ILLEGAL PRIORITY NUMBER (WNo., UNo., SNo.)	A	L	S	Blue
Cause	The priority numbering order within a unit is not correct.				
Action	The machining order within one unit has been reversed by the incorrect priority numbering. Change the priority numbers.				
463	PRIORITY No. OVERFLOW (, ,)	A	L	S	Blue
Cause	A priority number exceeding 99 has occurred because an attempt has been made to move a priority number(s) on the PROGRAM LAYOUT display.				
Action	Set priority numbers in the correct order, and then move the desired priority number(s).				
464	ILLEGAL ADDRESS INPUT (, ,)	A	L	S	Blue
Cause	An address not covered by the specifications has been designated during input of subprogram unit addresses on the PROGRAM (MAZATROL) display.				
Action	Check and correct the address. Check the specifications.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
465	EIA SHAPE DATA NOT FOUND (, ,)	A	L	S	Blue
Cause	Although an attempt has been made to draw a workpiece shape using the selected EIA/ISO program, shape data is not present in that program.				
Action	Review the program. If the program is that which has been obtained by EIA/ISO conversion output, change the setting of the bit 0 of parameter F89 to 1 and then carry out the converting operation once again.				
466	INCORRECT EIA SHAPE DATA (, ,)	A	L	S	Blue
Cause	Although an attempt has been made to draw a workpiece shape using the selected EIA/ISO program, the corresponding shape data is not correct.				
Action	Review the program.				
467	— (, ,)				
Cause	—				
Action	—				
468	MAINTENANCE CHECK WARNING (, ,)		L	S	Blue
Cause	The target time of the items which had been set on the MAINTENANCE CHECK display has been exceeded.				
Action	Carry out periodic checks, and then after completion of the checks, reset the current time of the check items of the MAINTENANCE CHECK display to zero (0).				
469	TPC DATA EDIT IMPOSSIBLE (, ,)	A	L	S	Blue
Cause	The TPC data setting is not possible for the designated unit.				
Action	Check the program.				
470	ILLEGAL TPC DATA (, ,)	B	L	S	Blue
Cause	The TPC data for the unit is not correct. After setting the TPC data, the unit machining mode has been changed.				
Action	Delete the TPC data and set correct TPC data once again whenever required.				
471	TPC DATA NOT FOUND (, ,)	A	L	S	Blue
Cause	While the cursor was on a line of unit not containing TPC data on the display, the [TPC] menu key was pressed during the program list mode.				
Action	Press the [TPC] menu key after shifting to the programming mode.				
472	CALCULATION ERROR (, ,)	B	L	S	Blue
Cause	1. The calculation expressions displayed in the desk calculator window includes expressions that result in a calculation failure. 2. Calculating tool lengths A and B has been attempted in spite of tool nose position storage not being completed.				
Action	1. Review the calculation expressions, and correct nonexecutable sections, such as those which may include division by 0 or result in a negative number in SQRT. 2. Store the position of the tool nose.				
473	FINAL POINT DATA NOT FOUND (, ,)	B	L	S	Blue
Cause	Although the end-point data of the preceding line was not yet set in the BAR or CPY unit, the [NEXT] menu key was pressed.				
Action	Press the [NEXT] menu key after setting the end-point data of the preceding line.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
474	NO PROGRAM DISPLAY (TAPE MODE) (, ,)	A	L	S	Blue
Cause	1. During tape operation mode, an attempt has been made to select the EIA MONITOR display. 2. An attempt has been made to change the operation mode to tape operation mode when the EIA MONITOR display is selected.				
Action	1. During tape operation mode, programs cannot be displayed on the EIA MONITOR display. 2. Select a display other than the EIA MONITOR display before changing the operation mode to tape operation mode.				
475	NO EIA/ISO OPTION (, ,)	A	L	S	Blue
Cause	An attempt has been made to use an EIA/ISO-option related function in spite of the absence of an EIA/ISO option.				
Action	An EIA/ISO-option related function cannot be used since the system has no EIA/ISO option.				
476	NO OPTION (, ,)	A	L	S	Blue
Cause	An attempt has been made to use an optional function in spite of the absence of that option.				
Action	This function cannot be used since the system does not have the option for the function.				
477	— (, ,)				
Cause	—				
Action	—				
478	MEMORY PROTECT (MEASURING) (, ,)	A	L	S	Blue
Cause	An attempt has been made to copy the coordinates data in WRITE DATA column of the MEASURE display into the designated position, while measurement using the MEASURE display is in progress.				
Action	Copy the coordinates data only after the measurement has been completed.				
479	— (, ,)				
Cause	—				
Action	—				
480	CARD NOT READY (, ,)				
Cause	Executing the memory card I/O function in DATA I/O display mode has been attempted when a memory card was not set.				
Action	Set a memory card before using the memory card I/O function. If a memory card has been set, confirm the insertion direction of the memory card and set it again. If the alarm recurs even so, the memory card is likely to be damaged; replace the memory card and re-execute the I/O function.				
481	DIRECTORY NOT FOUND (, ,)	A	L	S	Blue
Cause	Input/output operations on a directory not present in the memory card have been attempted during memory card I/O in DATA I/O display mode.				
Action	Check whether the specified directory is present.				
482	— (, ,)				
Cause	—				
Action	—				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
483	SOLID DESCRIPT. IMPOSSIBLE UNIT (WNo., UNo.,)	B	I	S	Blue
Cause	On the TOOL PATH CHECK display in the solid mode, an attempt was made to check the tool path in a program containing a unit or a condition inappropriate to solid display (program of initial point scheme or with POS-B in the indexing unit set at an angle other than 0°).				
Action	Before checking the tool path, cancel solid mode or delete the unit concerned.				
484	INCORRECT SPINDLE TOOL (, ,)	A	L	S	Blue
Cause	The [LENG-OFS TEACH] menu key has been pressed in spite of the fact that a tool not having its "Length Offset Data" item displayed in TOOL DATA display mode (for example, a turning tool) was mounted in the spindle.				
Action	Check the tool mounted in the spindle.				
485	INCORRECT HEAD INDEX (, ,)	A	L	S	Blue
Cause	Executing the tool nose position storage function (TEACH) has been attempted when the head index angle displayed in TOOL DATA display mode was neither 0°, 90°, nor 180°.				
Action	Execute the tool nose position storage function (TEACH) when the head index angle is either 0°, 90° or 180°.				
486	ILLEGAL JAW DATA (, ,)	A	L	S	Blue
Cause	The use of the jaw shape that has been specified for the program results in interference between adjacent jaws as a result of the SOLID mode PATH CHECK.				
Action	Review the jaw data or the workpiece-gripping diameter value.				
487	LONG BORING BAR ADAPTER EXISTS (Pocket No, ,)	A	L	S	Blue
Cause	Assigning a tool other than an inside-diameter turning tool to the magazine pocket holding the adapter for the long boring bar has been attempted.				
Action	Review the pocket number or the type of tool to be assigned.				
488	MEMORY PROTECT (MAGAZINE SETUP) (, ,)	A	L	S	Blue
Cause	Tool data updating operations (such as editing tool data, completing the layout of tools, or downloading data on the DATA I/O display) have been attempted during magazine setup.				
Action	Perform tool data updating operations after completing the magazine setup operations. Release the reverse display mode of the [MAGAZINE SETUP] menu item relating to visual tool data management.				
489	CANNOT SET THREAD POSITION (, ,)	A	L	S	Blue
Cause	An attempt was made to set jaw dimensions without having selected the jaw type (OUT1/IN/OUT2).				
Action	Select the jaw type beforehand.				
490	REGISTRATION NUMBER EXCEEDED (, ,)	A	L	S	Blue
Cause	The data registration has exceeded its maximum value available. An attempt was made on the TOOL FILE display to register the ninth data item of workpiece material.				
Action	Delete unnecessary items to register the desired one.				
491	ITEM NOT SETUP (, ,)	A	L	S	Blue
Cause	Data registration is not yet complete.				
Action	Check if all the data items are set as required.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
492	CANNOT EDIT 3D DATA (, ,)	B	L	S	Blue
Cause	Editing the CAD tool model has been attempted.				
Action	The CAD tool model cannot be edited.				
493	SELECT JAW TYPE (, ,)	A	L	S	Blue
Cause	Assigning dimensions to jaw data for which a shape has not been specified has been attempted.				
Action	Specify some shape before assigning dimensions.				
494	CANNOT SELECT TNO (NOT MAGAZINE) (, ,)	A	L	S	Blue
Cause	Execution of tool selection has been attempted under the selected status of a non-magazine type turret.				
Action	Select a magazine type of turret before executing tool selection.				
495	INCORRECT CAD MODEL DATA (Note, ,)	B	L	S	Blue
Cause	Registered tool model data contains imperfections (a tool model cannot be drawn with the entered data).				
Action	Correct the data in the corresponding section. Note : During Parametric model entry The alarm-causing section is displayed. During rotating/extruding tool model entry Causes are displayed. 11: The entered shape has overlapping elements. 12: The entered shape is clockwise (CW). 13: The line connecting the starting and ending points overlaps some other element.				
496	— (, ,)				
Cause	—				
Action	—				
497	HEAD ANGLE INCORRECT (, ,)	A	L	S	Blue
Cause	1. An attempt has been made to storage the tool tip position with the head not in its horizontal machining position (TOOL DATA or TOOL OFFSET display only). 2. An attempt has been made to perform MDI-MMS or manual measurement with the head not in its horizontal or vertical machining position. <HV>				
Action	1. Tool tip position storage (tool length measurement) is possible only with the head in its horizontal machining position. 2. MDI-MMS or manual measurement is possible only with the head in its horizontal or vertical machining position.				
498	NO HEAD DATA (, ,)	A	L	S	Blue
Cause	Offset data for the selected head is not registered on the HEAD OFFSET display.				
Action	Check if the selected head data is registered on the HEAD OFFSET display.				
499	ILLEGAL HEAD TYPE (, ,)	A	L	S	Blue
Cause	An attempt has been made to perform tool tip position storage on the TOOL DATA or TOOL OFFSET display during use of the horizontal type of head.				
Action	To perform tool tip position storage for tool length measurement, mount a vertical head or a cover.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1400	SYSTEM ERROR (, ,)	A	L	S	Blue
Cause	An internal trouble has occurred in the system.				
Action	Contact Mazak Technical Center or Technology Center with the information of the particular operation causing the alarm, and of the codes displayed in parentheses.				
1401	FILE OPEN ERROR (, ,)	A	L	S	Blue
Cause	Trouble has occurred in opening a file from the hard disk.				
Action	The file in question might be currently being accessed by another application. Wait for a while to repeat the same operation. If the alarm is not cleared, contact Mazak Technical Center or Technology Center with the information of the particular operation causing the alarm, and of the codes displayed in parentheses.				
1402	FILE READ ERROR (, ,)	A	L	S	Blue
Cause	Trouble has occurred in reading a file on the hard disk.				
Action	The file in question might be currently being accessed by another application. Wait for a while to repeat the same operation. If the alarm is not cleared, contact Mazak Technical Center or Technology Center with the information of the particular operation causing the alarm, and of the codes displayed in parentheses.				
1403	FILE WRITE ERROR (, ,)	A	L	S	Blue
Cause	Trouble has occurred in writing a file on the hard disk.				
Action	The file in question might be currently being accessed by another application. Wait for a while to repeat the same operation. If the alarm is not cleared, contact Mazak Technical Center or Technology Center with the information of the particular operation causing the alarm, and of the codes displayed in parentheses.				
1404	MEMORY PROTECT (DATA IN USE) (, ,)	A	L	S	Blue
Cause	It was attempted to edit jaw shape data of the machining program currently selected.				
Action	Change the selection of the program to edit the required data.				
1405	FILE SIZE ERROR (, ,)	A	L	S	Blue
Cause	The size of the loaded file is not correct.				
Action	The file in question might be destroyed. Contact Mazak Technical Center or Technology Center with the information of the particular operation causing the alarm, and of the codes displayed in parentheses.				
1406	LOW MEMORY (, ,)	A	L	S	Blue
Cause	Memory size currently available is too small.				
Action	Exit all the unnecessary applications to make the best of the NC ones. If the alarm is caused again, contact Mazak Technical Center or Technology Center with the information of the particular operation causing the alarm, and of the codes displayed in parentheses.				
1407	FILE CREATION ERROR (, ,)	A	L	S	Blue
Cause	Creating a file and folder on the hard disk was not successful.				
Action	Contact Mazak Technical Center or Technology Center with the information of the particular operation causing the alarm, and of the codes displayed in parentheses.				
1408	DATA TRANSFER ERROR (, ,)	A	L	S	Blue
Cause	Data transfer to the NC was not successful.				
Action	Contact Mazak Technical Center or Technology Center with the information of the particular operation causing the alarm, and of the codes displayed in parentheses.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1409	MILLING ONLY SELECTED (, ,)	A	L	S	Blue
Cause	The designated function is not available for a program of initial-point scheme (ONLY MILL).				
Action	Use the function for a program of another scheme.				
1410	MILL & TURN (WPC) SELECTED (, ,)	A	L	S	Blue
Cause	The designated function is not available for a program of workpiece scheme (MILL & TURN) with WPC setting.				
Action	Use the function for a program of another scheme.				
1411	MILL & TRN (Z-OFS) SELECTED (, ,)	A	L	S	Blue
Cause	The designated function is not available for a program of workpiece scheme (MILL & TURN) with Z-offset setting.				
Action	Use the function for a program of another scheme.				
1412	INCORRECT G-CODE (, ,)	A	L	S	Blue
Cause	The designated function is not available for the G-code system selected by the F30 parameter.				
Action	Correct the program, or change the F30 setting.				
1413	INCORRECT RUNNING MODE (, ,)	A	L	S	Blue
Cause	The designated function is not available for the current operation mode or on the current display.				
Action	Change the operation mode or display.				
1414	PROGRAM WRITING (, ,)	A	L	S	Blue
Cause	Storing a program on the hard disk was not successful.				
Action	The file in question might be currently being accessed by another application. Wait for a while to repeat the same operation. If the alarm is not cleared, contact Mazak Technical Center or Technology Center with the information of the particular operation causing the alarm, and of the codes displayed in parentheses.				
1415	ZERO CALCULATED TOOL LENGTH (, ,)	A	L	S	Blue
Cause	The LENGTH A item of the tool is set to zero (0).				
Action	Check the settings of tool length or tool set data and, if required, measure the tool in question again.				
1416	CANNOT INDEX TOOL (, ,)	A	L	S	Blue
Cause	The tool currently indexed on the turret is not appropriate.				
Action	Index an appropriate tool beforehand.				
1417	PARAMETER ERROR (, ,)	A	L	S	Blue
Cause	The setting in a parameter is not correct.				
Action	Check and correct the parameter setting with reference to the codes displayed in parentheses.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1418	CANNOT USE LOWER TURRET (, ,)	A	L	S	Blue
Cause	The current selection of the headstock to which the lower turret is subordinate is not correct.				
Action	Change over the headstock selection concerned.				
1419	— (, ,)				
Cause	—				
Action	—				
1420	FILE SIZE LIMIT EXCEEDED (, ,)	A	L	S	Blue
Cause	The size of the file is too large for new data to be written in it.				
Action	Delete the file, or rename the file.				
1421	— (, ,)				
Cause	—				
Action	—				
1422	PROGRAM LAYOUT ERROR (, ,)	A	L	S	Blue
Cause	Creating the data for turret-by-turret display of program layout was not successful.				
Action	Change the display mode.				
1423	TOOL PATH CHECK NOT COMPLETE (, ,)	A	L	S	Blue
Cause	Since the program has not yet undergone tool path check, the VISUAL TOOL MANAGEMENT display cannot perform calculations for indicating the spare tools to be additionally provided for the machining operation with the program.				
Action	Execute the function for tool path check on the program.				
1424	BARRIER INFORMATION NOT SET (, ,)	A	L	S	Blue
Cause	In the program-listing mode the [BARRIER INFORM.] menu function is selected without any barrier data being registered.				
Action	Select the program-creating mode before pressing the [BARRIER INFORM.] menu key.				
1425	ILLEGAL SETUP DATA (, ,)	A	L	S	Blue
Cause	The setup data are not correct.				
Action	Make a copy of the program in question on the PROGRAM FILE display, and prepare the setup data for the program copied.				
1426	NO PROG LAYOUT FOR THIS PROCESS (, ,)	A	L	S	Blue
Cause	The current process has no program layout information provided.				
Action	Press the menu selector key to select a process with program layout data, or select the display mode for all processes. Alternatively, select a process with program layout data on the PROGRAM (MAZATROL) display or PROCESS CONTROL display and then call up the PROGRAM LAYOUT display from there.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1427	3D INTERFRNCE CHECK MODEL ERROR (, ,)	A	L	S	Blue
Cause	3D remodeling of the workpiece, fixture, or tool has failed.				
Action	Modify the entered data. Confirm the model.				
1428	INCORRECT POCKET FOR LONG BOR. BAR (, ,)	A	L	S	Blue
Cause	A long boring bar is assigned to a position other than the special pocket for long boring bar when the [LAYOUT FINISH] menu key is pressed on the VISUAL TOOL MANAGEMENT display.				
Action	Assign the long boring bar to the special pocket.				
1429	CANNOT PERFORM AUTO SET (, ,)	A	L	S	Blue
Cause	Tool setup auto-setting for a tool not capable of using the tool setup auto-setting function (e.g., a magazine tool or a tool facing the No. 1 turning spindle) has been attempted on the TOOL DATA display.				
Action	Apply the tool setup auto-setting function only to a turret tool facing the No. 2 turning spindle.				
1430	NOT ACTIVE PROGRAM (, ,)	A	L	S	Blue
Cause	This alarm indicates that the machine-operating program and the program selected on the PROGRAM (MAZATROL) display differ during execution of the workpiece transfer storage function on the PROGRAM (MAZATROL) display.				
Action	Match the machine-operating program and the program selected on the PROGRAM (MAZATROL) display.				
1431	AUTO SET ERROR (, ,)	A	L	S	Blue
Cause	The tool for which the reference tool length was set has not been found during automatic tool length setting.				
Action	Set at least one tool length value before executing automatic tool length setting.				
1432	TOOLMODEL NOT SELECTED (, ,)	A	L	S	Blue
Cause	The tool model to be used as a reference has not been selected during tool model copying, erasing, or renaming.				
Action	Select a reference tool model before executing tool model copying, erasing, or renaming.				
1433	SAME TOOL MODEL NAME IN USE (, ,)	A	L	S	Blue
Cause	A registered tool name has been specified during tool model copying or renaming.				
Action	Specify a tool name that has not been registered.				
1434	INCOR CRS POS FOR PROG LAYOUT (, ,)	A	L	S	Blue
Cause	When setting the PROGRAM LAYOUT display from the MAZATROL program was attempted, the layout of program data up to the first end unit has been displayed since an end unit was present at a position previous to the cursor.				
Action	Move the cursor to a position previous to the end unit, and then set the PROGRAM LAYOUT display.				
1435	3D MONITOR MANUAL MODE ONLY (, ,)	A	L	S	Blue
Cause	Setting the 3D MONITOR display has been attempted during automatic operation mode.				
Action	Select manual operation mode before setting the 3D MONITOR display.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1436	DIFFERENT TYPE OF PROGRAM (, ,)	A	L	S	Blue
Cause	A program of a different program type has been specified during MAZATROL program copying.				
Action	Specify a program of the same type.				
1437	CANNOT TRANSFER ACTIVE PROGRAM (, ,)	A	L	S	Blue
Cause	Transferring the same program as the automatic operation program, from the backup area to the standard area, has been attempted.				
Action	Before conducting program transfer, select a different program for automatic operation.				
1438	NO SETUP DATA (, ,)	A	L	S	Blue
Cause	The [SETUP DATA] menu key has been pressed in program listing mode when there was no registered setup data.				
Action	Select programming mode before pressing the [SETUP DATA] menu key.				
1439	CANNOT CHANGE UNIT (, ,)				
Cause	Changing a unit has been attempted when a unit change in the MAZATROL program was prohibited. <ul style="list-style-type: none"> • It is not allowed to change a unit for which data other than the unit name has been set up. • It is not allowed to change a unit if the current machining section is not selectable for the new unit (e.g., a unit for which an outside-diametral section has been selected as the machining section cannot be changed to a turning-drilling unit). 				
Action	Review the program.				
1440	ID No. NOT FOUND (, ,)				
Cause	Although moving tool data from the magazine tool list to the stored-tool list has been attempted on the VISUAL TOOL MANAGEMENT display, an ID number has not been set up for the corresponding tool.				
Action	Set the ID number in the tool data.				
1441	FILE NOT FOUND (, ,)				
Cause	The corresponding file does not exist during file loading on the MAINTENANCE CHECK display.				
Action	Create "maintenance.txt" under "C:\nm64mdata\language-name".				
1442	ILLEGAL TOOL- CANNOT ASSIGN (, ,)	A	L	S	Blue
Cause	Registering an illegal tool in tool data has been attempted.				
Action	Confirm the type of tool to be registered.				
1443	BACKUP INCOMPLETE (, ,)	A	L	S	Blue
Cause	NC data backup has been aborted.				
Action	To conduct the backup, perform the backup operations once again.				
1444	PROCESS TOO LARGE -DEVIDE PROC (, ,)	A	L	S	Blue
Cause	Assigning a priority number to the roughing tool on the PROGRAM LAYOUT display has been attempted, but the priority numbers required exceed 99. The priority number must range from 1 to 99. Otherwise, this alarm will occur.				
Action	Split the process and execute the [ROUGH PRIORITY] menu function.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1445	SAME SIMULTAN. NO IN RGH & FIN (, ,)	A	L	S	Blue
Cause	Assigning a priority number to the roughing tool on the PROGRAM LAYOUT display has been attempted, but since simultaneous machining is specified in the program and the same simultaneous machining number is specified for both roughing and finishing. Roughing cannot be prioritized with the simultaneous machining number ignored.				
Action	Review the specified simultaneous machining number and execute the [ROUGH PRIORITY] menu function.				
1446	CANNOT CHNG.WNo (SNGL PROC.MODE) (, ,)	A	L	S	Blue
Cause	A work number change has been attempted in single-process mode.				
Action	Cancel the single-process mode before conducting the work number change.				
1447	RESTORE OPERATION STOPED (, ,)	A	L	S	Blue
Cause	The NC data restore process has been aborted.				
Action	To conduct the restore process, perform the restore operations once again.				
1448	TOOL LENGTH OVER (, ,)	A	L	S	Blue
Cause	The tool length input range has been overstepped as a result of tool measurement.				
Action	Use a tool of any tool length value falling within the range defined in parameters SU110 - SU113 .				
1449	NC PROGRAM MEMORY SIZE OVER (AREA, ,)	A	L	S	Blue
Cause	This alarm indicates that during hard-disk program transfer to the program storage area (NC memory), as in a power-on status, lack of the necessary NC memory space has been detected or the program storage capacity limit exceeded. If AREA = 1 indicates alarm during program transfer from the standard area to the NC memory If AREA = 2 indicates alarm during machine manufacturer macroprogram transfer to the NC memory If AREA = 3 indicates alarm during GL pattern file transfer to the NC memory				
Action	Use the PROGRAM FILE display to copy or move programs.				
1450	SELECT LOWER TURRET (, ,)	A	L	S	Blue
Cause	An attempt has been made to use the [TEACH] menu function for a WPC unit in spite of the lower turret being selected.				
Action	The [TEACH] menu function cannot be used for a WPC unit with the lower turret being selected. Designate the upper turret.				
	(, ,)				
Cause					
Action					
1455	ILLEGAL MAKER NAME (, ,)	A	L	S	Blue
Cause	Invalid characters have been used in the manufacturer name in an attempt to add a tool part model.				
Action	The characters \ / : * ? “ < > cannot be used in the manufacturer name. Use other characters to assign the manufacturer name.				
1456	ILLEGAL MODEL NO (, ,)	A	L	S	Blue
Cause	Invalid characters have been used in the model number in an attempt to add a tool part model.				
Action	The characters \ / : * ? “ < > cannot be used in the model number. Use other characters to assign the model number.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1457	CANNOT EDIT MAKER MADE DATA (, ,)	A	L	S	Blue
Cause	Editing or deleting the tool part model that the manufacturer created has been attempted.				
Action	The tool part model that the manufacturer created cannot be edited or deleted.				
1458	— (, ,)				
Cause	—				
Action	—				
1459	— (, ,)				
Cause	—				
Action	—				
1460	CANNOT USE V.M. DURING AUTO OPE (, ,)	A	L	S	Blue
Cause	Calling up the VIRTUAL MACHINING display was attempted during automatic operation (when the INTELLIGENT SAFETY SHIELD function in automatic operation mode was valid).				
Action	When the INTELLIGENT SAFETY SHIELD function in automatic operation mode is valid, the VIRTUAL MACHINING display cannot be presented during automatic operation. Change the display mode after automatic operation has been completed or after making the INTELLIGENT SAFETY SHIELD function in automatic operation mode invalid.				
1461	TOOL MODEL IMPORT FAILURE (, ,)	A	L	S	Blue
Cause	Importing the CAD tool model has failed.				
Action	The data to be imported is incorrect or importable data is absent at the specified location. Make sure that the data to be imported is correct or that data is saved in the specified path.				
	(, ,)				
Cause					
Action					
1470	INCORRECT DESIGNATED DATA (, , Note)	B	L	S	Blue
Cause	<p>Note :</p> <ul style="list-style-type: none"> -1 : Data in the error file for space error compensation to be transferred to the NC unit does not match the machine configuration. -2 : Error components in the error file for space error compensation have exceeded the specification (single direction: 128 points, both directions: 64 points). -3 : Data in the error file for space error compensation is illegal. 				
Action	<p>Take action below against respective causes described above.</p> <ul style="list-style-type: none"> -1 : Input the data measured on the corresponding machine. -2 : Make measurements again so that the number of measurement points for each error component will not be greater than 128 points (single direction) or 64 points (both directions). -3 : Match the file format (single direction/both directions) of error data with the valid mode (single direction/both directions). <p>If the problem is not solved even after the action above, contact Mazak Technical Center or Technology Center.</p>				
1471	1BLOCK SIZE OVER (, ,)	A	L	S	Blue
Cause	The G code macro call output from the add-in EIA window has exceeded 248 characters.				
Action	Change the argument setting to make it within 248 characters.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1472	INVALID REEDIT (, ,)	A	L	S	Blue
Cause	<p>The [RE-EDIT] menu key is pressed when the block in the cursor position cannot be edited in the EIA/ISO program.</p> <ol style="list-style-type: none"> Unsuitable G code or unnecessary address for add-in EIA function is included in the block in the cursor position. A comment is located toward the top of the program from the block of G code macro call. The block for re-editing exceeds 248 characters. 				
Action	<ol style="list-style-type: none"> Delete unnecessary G codes and addresses from the block. For blocks to be re-edited with the add-in EIA function, the comment must be added at the end of the block. Edit the block to include 248 characters or less. 				
1473	NOT AUTO MODE (, ,)	A	L	S	Blue
Cause	The restart 2 non-modal function setting was attempted in a mode other than the memory operation or hard disk operation mode.				
Action	Perform the restart 2 non-modal function setting in the memory operation or hard disk operation mode.				
	(, ,)				
Cause					
Action					
1477	INCORRECT PROCESS SELECTION (, ,)	A	L	S	Blue
Cause	<p>A process for a head differing from the selected head has been selected in the SELECT PROCESS window for the restart or single process operation.</p> <p>(Only for MULTIPLEX series)</p>				
Action	<p>Select a process for the selected head.</p> <p>It is also possible to avoid the alarm by setting parameter F158 bit5 = 0.</p>				
	(, ,)				
Cause					
Action					
1482	CANNOT LOCK (LIMIT NUMBER) (, ,)	A	L	S	Blue
Cause	With nine messages locked, an attempt to lock a 10th message was made.				
Action	The number of messages to be locked must be nine or smaller.				
	(, ,)				
Cause					
Action					

3-2-6 No. 500 - No. 599, No. 1500 - No. 1599 (I/O error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display
500	— (, ,)				
Cause	—				
Action	—				
501	ILLEGAL FORMAT (, ,)	A	L	S	Blue
Cause	An external storage medium that contains data other than MATRIX data has been set.				
Action	Check the contents of the external storage medium for appropriateness to the MATRIX.				
502	CANNOT LOAD (PROG SIZE EXCEED) (WNo., ,)	A	L	S	Blue
Cause	The contents of the external storage medium are not correct. (Loading of a MAZATROL program of more than 2000 lines of data has been attempted.)				
Action	Either use another external storage medium, or save the program data once again. After that, carry out the load operation once again.				
503	CANNOT LOAD (TOO MANY PROGRAMS) (WNo., ,)	A	L	S	Blue
Cause	An attempt has been made to load more machining programs than the maximum number of programs that can be registered within the NC unit.				
Action	Delete unnecessary programs, or save the programs onto an external storage and then delete them. After that, load the particular program.				
504	CANNOT LOAD (AUTO OPERATION) (, ,)	A	L	S	Blue
Cause	An attempt has been made during automatic operation to load data other than machining programs.				
Action	Load the data only after completion of automatic operation.				
505	CANNOT LOAD (MISMATCH) (, ,)	A	L	S	Blue
Cause	Loading has been attempted although the data within the external storage medium does not match to the NC unit (Mismatching in data size, etc.).				
Action	Check if the data saved on the external storage medium is the data to be used for the machine currently in operation.				
506	SAME PROGRAM No. DESIGNATED (WNo., ,)	A	L	S	Blue
Cause	An attempt has been made to load the machining program that has the same work number as that of a machining program registered within the NC unit.				
Action	Check for an overlapping work numbers.				
507	NO DESIGNATED PROGRAM (WNo., ,)	A	L	S	Blue
Cause	The machining program whose saving has been attempted does not exist in the NC unit.				
Action	Check if the machining program with the specified work number exists in the NC unit.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
508	MEMORY CAPACITY EXCEEDED (WNo., ,)	A (G)	L (L)	S (S)	Blue (Blue)
Cause	1. An attempt has been made to load machining program data that exceeds the maximum available area for program registration within the NC unit. 2. In the middle of saving onto the external storage medium, data saving exceeding the memory capacity has been made, or the end-of-tape (or end-of-disk) code has been detected.				
Action	1. Delete unnecessary programs, or save the programs onto an external storage and then delete them. After that, load the particular program. 2. Split the data into segments according to the particular size of the free saving area within the external storage medium, and then carry out the saving operations once again.				
509	MEMORY PROTECT (, ,)	A	L	S	Blue
Cause	Loading has been attempted when the PROGRAM LOCK/ENABLE switch setting was LOCK.				
Action	Set the switch to ENABLE, and then carry out the loading operation.				
510	DATA DO NOT MATCH (WNo., ,)	A	L	S	Blue
Cause	Comparison between the external storage medium contents and the NC memory contents has shown disparities in data size, type of file information, etc.				
Action	1. Locate those disparities on the PROGRAM FILE display and correct them, and then make the comparison once again. 2. If the disparities exist in data other than machining program data, check if the data is for the machine being used.				
511	PROGRAM DATA NOT SAME (WNo., UNo., SNo.)	A	L	S	Blue
Cause	Comparison between the cassette tape or floppy disk contents and the NC data has shown several disparities.				
Action	1. After correcting the disparities within the machining program, make the comparison once again. 2. If the disparities exist in data other than machining program data, locate those disparities on each display.				
Note : This alarm message may be displayed if data is saved prior to automatic operation and then subjected to comparison with that after automatic operation. This is because execution of automatic operation may cause automatic data overriding.					
512	NO EIA/ISO OPTION (WNo., ,)	A	L	S	Blue
Cause	An attempt has been made to load an EIA/ISO program in spite of the absence of an EIA/ISO option.				
Action	An EIA/ISO program cannot be loaded since the system has no EIA/ISO option.				
513	PROGRAM DATA TYPE INCORRECT (, ,)	A	L	S	Blue
Cause	An attempt has been made to load a machining program different in structure from the programs within the NC memory.				
Action	Check the contents of the external storage medium for appropriateness to the MATRIX.				
514	DATA TYPE INCORRECT (, ,)	A	L	S	Blue
Cause	An attempt has been made to load data (other than machining program data) that differs in structure from the NC memory data.				
Action	Check the contents of the external storage medium for appropriateness to the MATRIX or the machine being used.				
515	INCORRECT DESIGNATED DATA (, ,)	A	L	S	Blue
Cause	1. During I/O operation with a memory card, an attempt has been made to load data the structure of which is not correct. 2. During I/O operation with a floppy disk, an attempt has been made to load data the structure of which is not correct.				
Action	1. Check if the data saved during I/O operation with a memory card is for MATRIX. 2. Check if the data saved during I/O operation with a floppy disk is for MATRIX.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
516	SYSTEM ERROR (, ,)	E	L	S	Blue
Cause	1. When program loading was attempted, there was not a comment file (C:\MC_MachinePrograms\index.tbl). 2. An error has occurred within the system.				
Action	1. Enter any comment on the PROGRAM FILE display, and load the program. Do not edit or delete "index.tbl" since it is the management file for NC. 2. Contact Mazak Technical Center or Technology Center. (At this time, also please notify them of what kind of operating procedure you had carried out before the alarm message appeared and what values were displayed in parentheses.)				
517	PROG. OPERATION NOT ALLOWED (WNo., ,)	A	L	S	Blue
Cause	1. An attempt has been made to save a display inhibiting program. (Program management function) 2. An attempt has been made to save the program being edited or the program being loaded using another I/O unit.				
Action	1. Check if the specified work number is for the program of display inhibition. 2. Carry out a saving operation only after completion of the program editing operation (or the program loading operation using another I/O unit).				
518	DATA OPERATION NOT ALLOWED (, ,)	A	L	S	Blue
Cause	1. An attempt has been made during automatic operation to load data other than machining program data. 2. An attempt has been made to save the data being loaded using another I/O unit. 3. An attempt has been made to load the data being saved using another I/O unit.				
Action	Wait until automatic operation has been completed (or until the loading or saving operation using another I/O unit has been completed).				
519	DATA SIZE EXCEEDED (WNo., Note,)	A	L	S	Blue
Cause	The EIA/ISO machining program includes a block that consists of more than 256 characters. (EOB or EOR does not appear within 256 characters.)				
Note :	The number displayed next to the work number is a line number, which corresponds to the number displayed in the lower right section of the PROGRAM display.				
Action	Correct the EIA/ISO machining program. (Insert EOB within 256 characters.)				
520	EIA/ISO CONVERT ERROR (WNo., ,)	B	L	S	Blue
Cause	Nonconvertible sections have been found when an attempt was made to convert the MAZATROL program into an EIA/ISO program.				
Action	Review the MAZATROL program.				
521	CANNOT LOAD (WRONG PROG TYPE) (, ,)	A	L	S	Blue
Cause	The operator has attempted loading a machining program of a scheme different from the program scheme within the NC unit.				
Action	Confirm the program scheme of the machining program.				
522	SAME SET No. EXISTS (, ,)	B	L	S	Blue
Cause	An attempt has been made to load the GL setup data that has the same GL setup number as that of setup data registered within the NC unit.				
Action	Rename or erase the GL setup data within the NC unit before loading the data.				
523	IMPOSSIBLE TO CONVERT EIA (Prg. No. Processing No., ,)	B	L	S	Blue
Cause	1. The parameter file does not exist under the same directory as that of the program. 2. The G-code is other than standard G-code Series A. 3. The program is a subprogram.				
Action	1. Save the parameter file under the same directory as that of the program. 2. The G-code cannot be converted since it is not supported for MATRIX. 3. The program cannot be converted since the system concerned is unknown.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
524	— (, ,)				
Cause	—				
Action	—				
525	HDD I/O ERROR (Cause, ,)				
Cause	An error has occurred during output of data of the measurements results print-out function to the hard disk drive.				
Action	Check if the available space within the hard disk drive is sufficient for the purpose.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
526	PROFIBUS-DP COMMUNICATION ERROR (Note 1, Note 2,)	E	L	M, O, Q	Blue
	<p>Communication error has occurred during the PROFIBUS-DP communication using a field network expansion card.</p> <p>Note :</p> <ol style="list-style-type: none"> 1. <p></p> <p>Global error bit Indicates the error status (hexadecimal).</p> <ul style="list-style-type: none"> 01: Control error (parameter error) 02: Auto clear error (Since an update at a slave station has failed, updates at all slave stations have stopped.) 04: No-exchange error (a slave station has an update error) 08: Critical error (communication cannot continue due to a serious network problem) 10: Not ready (NC communication is not ready) 20: Timeout error (a timeout is detected during an update with each station) <p>Master communication status Indicates the status of the master station (hexadecimal).</p> <ul style="list-style-type: none"> 00: Off-line (being initialized) 40: Stop (I/O stopped) 80: Clear (resetting the output data from each slave station by sending "0" data) C0: Running (I/O being updated) 2. <p></p> <p>Slave communication status Indicates the communication status of the slave station at which a communication error is occurring (hexadecimal).</p> <ul style="list-style-type: none"> 00: The target slave exists on the bus. An appropriate data communication is being made between the master station and the target slave station. 01: No data communication is being made between the master station and the target slave station. Configuration with the target slave station may not be finished correctly. There may be no response from the target slave station. 02: No data communication may be being made between the master station and the target slave station. Diagnosis information from the target slave station may exist in the internal buffer of the master station. 03: The target slave exists on the bus. An appropriate data communication is being made between the master station and the target slave station. Diagnosis information from the target slave station may exist in the internal buffer of the master station. <p>Slave station number Indicates the number of the slave station at which a communication error is occurring (hexadecimal).</p> <p>However, if no communication is being made with the master station, "00" is displayed.</p> <ul style="list-style-type: none"> 00: No communication is being made with the master station. 01: Station No. 1 02: Station No. 2 ⋮ 7D: Station No. 125 				
Action	Check the environment of the master station or slave station at which a communication error is occurring.				

3

ALARM

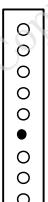
No.	Message	Type of error	Stopped status	Clearing procedure	Display																		
527	ETHERNET/IP COMMUNICATION ERROR (Note 1, Note 2,)	E	L	M, O, Q	Blue																		
	1. Communication error has occurred during the EtherNet/IP communication using a field network expansion card. 2. EtherNet/IP connection wait time setting may not be appropriate.																						
	Note : 1.																						
	 <p>Adaptor status</p> <p>0x00: Scanner stopped or connection failed 0x01: Communication being made 0x02: Communication stopped (communication with at least one adaptor station has stopped)</p>  <p>Scanner status</p> <p>0x01: Configuration not finished 0x02: Stopped 0x03: Idling 0x04: Running</p>																						
	2.																						
	 <p>Communication error details</p> <table> <tbody> <tr> <td>0x0000: Scanner stopped or connection failed</td> <td>0x0124: Parameter error</td> </tr> <tr> <td>0x000c: Watchdog timeout</td> <td>0x0125: Illegal network address</td> </tr> <tr> <td>0x0100: Initialization failed</td> <td>0x0126: No security memory</td> </tr> <tr> <td>0x0101: Database access failed</td> <td>0x0140: Network failure</td> </tr> <tr> <td>0x0119: Configuration not executed</td> <td>0x0141: Communication failure</td> </tr> <tr> <td>0x0120: Configuration failed</td> <td>0x0142: Communication timeout</td> </tr> <tr> <td>0x0121: Data mismatch</td> <td>0x0143: Network isolation</td> </tr> <tr> <td>0x0122: Data discrepancy</td> <td>0x0144: Node redundancy</td> </tr> <tr> <td>0x0123: Inadequate data license</td> <td>0x0145: Cable not connected</td> </tr> </tbody> </table>					0x0000: Scanner stopped or connection failed	0x0124: Parameter error	0x000c: Watchdog timeout	0x0125: Illegal network address	0x0100: Initialization failed	0x0126: No security memory	0x0101: Database access failed	0x0140: Network failure	0x0119: Configuration not executed	0x0141: Communication failure	0x0120: Configuration failed	0x0142: Communication timeout	0x0121: Data mismatch	0x0143: Network isolation	0x0122: Data discrepancy	0x0144: Node redundancy	0x0123: Inadequate data license	0x0145: Cable not connected
0x0000: Scanner stopped or connection failed	0x0124: Parameter error																						
0x000c: Watchdog timeout	0x0125: Illegal network address																						
0x0100: Initialization failed	0x0126: No security memory																						
0x0101: Database access failed	0x0140: Network failure																						
0x0119: Configuration not executed	0x0141: Communication failure																						
0x0120: Configuration failed	0x0142: Communication timeout																						
0x0121: Data mismatch	0x0143: Network isolation																						
0x0122: Data discrepancy	0x0144: Node redundancy																						
0x0123: Inadequate data license	0x0145: Cable not connected																						
Action	1. Check the environment of the scanner or adaptor at which a communication error is occurring. 2. Review the EtherNet/IP connection wait time setting, and turn power off and then back on.																						

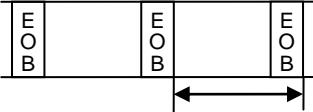
No.	Message	Type of error	Stopped status	Clearing procedure	Display
528	FIELDNETWORK SETTING ERROR (Note 1, Note 2,)	E	L	M	Blue
Cause	Note : <ol style="list-style-type: none"> 1. <p>Cause</p> <ul style="list-style-type: none"> 1: The value at R19974 (field network output data top register) is out of the setting range. 2: The value at R19976 (field network input data top register) is out of the setting range. 3: The value specified at the register R for high-speed data input/output is out of the setting range. 4: The value of parameter F150 is out of the setting range. 2. <p>Field network expansion card type</p> <ul style="list-style-type: none"> 0: Not connected 1: PROFIBUS-DP unit 2: EtherNet/IP unit 				
Action	<ol style="list-style-type: none"> 1. Review the value at R19974 (field network output data top register). 2. Review the value at R19976 (field network input data top register). 3. Review the value at the register R for high-speed data input/output. 4. Review the value of parameter F150. 				
529	— (, ,)				
Cause	—				
Action	—				
530	CMT MIS-CONNECTED (, ,)	G	L	S	Blue
Cause	<p>This message implies incorrect cable connection between CMT (cassette magnetic tape unit) or microdisk unit and the NC unit, or implies a power-off status or an incorrect baud-rate setting.</p> <p>In the case of microdisk unit, this message also implies incorrect setting of a floppy disk.</p>				
Action	<ol style="list-style-type: none"> 1. Check for correct cable connections. 2. Check if power is turned on. 3. Check for correct baud-rate setting. (Parameter for the NC unit: Baud rate) 4. For microdisk unit, check if the floppy disk is correctly set. 				
531	DESIGNATED FILE NOT FOUND (WNo., ,)	A	L	S	Blue
Cause	The machining program or another data that has been designated for the LOAD or COMPARE operation does not exist within the cassette tape or floppy disk.				
Action	Carry out a DIRECTORY operation to check what type of data is stored on the cassette tape or floppy disk.				
532	CMT NOT CONNECTED (, ,)	A	L	S	Blue
Cause	A cassette tape or floppy disk drive has not been mounted.				
Action	Correctly mount a cassette tape or floppy disk drive.				
533	NO OPERABLE DATA IN CMT (, ,)	A	L	S	Blue
Cause	The current M640 series-use cassette tape or floppy disk does not contain a saved machining program (only machining programs can be loaded from M640 series-use cassette tape or floppy disk).				
Action	<p>The cassette tape or floppy disk that has been registered for M640 series use does not contain a registered machining program.</p> <p>Perform checks using the M640 series NC unit.</p>				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
534	CMT I/O ERROR (, ,)	G	L	S	Blue
Cause	A hardware error has occurred in the CMT or microdisk unit.				
Action	Check the CMT or microdisk unit baud rate setting (RS-232C setting parameter), and replace the cassette tape or floppy disk.				
535	CMT WRITE PROTECT (, ,)	A	L	S	Blue
Cause	Data saving onto a write-protected cassette tape or floppy disk has been attempted.				
Action	The cassette tape or floppy disk is protected against data writing. Release the write-protected state. (For cassette tape, fill in the hole on the tape surface with tape.)				
536	POWER OFF DURING CMT OPERATION (, ,)	A	L	S	Blue
Cause	Power has been turned off during operation of the CMT or microdisk unit.				
Action	Check the machining program being transferred. If an anomaly is found, repeat the desired operation. If this alarm state has occurred during loading of a machining program, erase the loaded portion of the program and then execute the loading again.				
537	CMT MALFUNCTION (, ,)	G	L	S	Blue
Cause	Data cannot be read because of the presence of check sum errors, for example, within the cassette tape or floppy disk contents.				
Action	Reread the data only after setting a new cassette tape or floppy disk or after saving the corresponding data.				
538	— (, ,)				
Cause	—				
Action	—				
539	— (, ,)				
Cause	—				
Action	—				
540	TAPE READER MIS-CONNECTED (, ,)	G	L	S	Blue
Cause	This message implies incorrect cable connection between tape reader or microdisk unit and the NC unit or implies a power-off state. In the case of microdisk unit, this message also implies incorrect setting of a floppy disk.				
Action	1. Check for correct cable connections. 2. Check if power is turned on. 3. In the case of microdisk unit, check if the floppy disk is correctly set.				
541	TAPE PUNCHER MIS-CONNECTED (, ,)	G	L	S	Blue
Cause	This message implies incorrect cable connection between tape puncher or microdisk unit and the NC unit or implies a power-off state. In the case of microdisk unit, this message also implies incorrect setting of a floppy disk.				
Action	1. Check for correct cable connections. 2. Check if power is turned on. 3. In the case of microdisk unit, check if the floppy disk is correctly set.				
542	NO TAPE READER PUNCHER OPTION (, ,)	A	L	S	Blue
Cause	An attempt has been made to carry out a tape I/O operation although the tape reader/puncher option is not provided.				
Action	Provide the NC unit with a tape reader/puncher option. (Only with this option, tape I/O operations can be carried out.)				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
543	WNo. NOT FOUND ON PAPER TAPE (, ,)	A	L	S	Blue
Cause	Loading or comparing is not possible since no O numbers (work numbers) are stored on the paper tape or floppy disk.				
Action	Call the DATA I/O display (TAPE) and designate a work number(s).				
544	SET NEW PAPER TAPE (, ,)	A	L	S	Blue
Cause	1. The tape reader/puncher is not correctly loaded with paper tape. 2. Differences in baud-rate or other parameter settings for RS-232C exist between the tape reader/puncher (or microdisk unit) and the NC unit.				
Action	1. Check if the tape reader/puncher is correctly loaded with paper tape. 2. Check for differences in RS-232C parameter settings between the I/O unit and the NC unit.				
545	POWER OFF TAPE READ/PUNCH OPER. (, ,)	A	L	S	Blue
Cause	Power has been turned off during operation of the tape reader/puncher or microdisk unit.				
Action	If power has been turned off during loading, check the machining program loaded. If an error(s) is found, delete the loaded data and then reload the program. If power has been turned off during punching, re-punch the tape.				
546	TAPE READER ERROR (, ,)	G	L	S	Blue
Cause	A hardware error has occurred in the tape reader or the microdisk unit.				
Action	Before operating the tape reader or microdisk unit, check that no differences in RS-232C parameter settings exist between the tape reader or microdisk unit and the NC unit and replace the paper tape or floppy disk.				
547	TAPE PUNCHER ERROR (, ,)	G	L	S	Blue
Cause	A hardware error has occurred in the tape puncher or the microdisk unit.				
Action	Before operating the tape puncher or microdisk unit, check that no differences in RS-232C parameter settings exist between the tape puncher or microdisk unit and the NC unit and replace the paper tape or floppy disk.				
548	MAZATROL PROGRAM DESIGNATED (, ,)	A	L	S	Blue
Cause	An attempt has been made to punch a MAZATROL program onto paper tape.				
Action	Designate an EIA/ISO program. (Only EIA/ISO programs can be punched on paper tape.)				
549	DESIGNATED DATA NOT FOUND (, ,)	A	L	S	Blue
Cause	The designated data was not found on the paper tape or floppy disk.				
Action	Select another set of data or make a search once again from the beginning of the paper tape or floppy disk.				
550	PARITY H ERROR (, ,)	A	L	S	Blue
Cause	 The contents of the paper tape or floppy disk cannot be read since they include a parity-H error(s). (The number of holes on tape must always be even for ISO; it must be odd for EIA).				
Action	Reading must be carried out only after replacing the paper tape or floppy disk or after re-punching the program.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
551	PARITY V ERROR (, ,)	A	L	S	Blue
Cause	<p>The contents of the paper tape or floppy disk cannot be read since they include a parity-V error(s).</p>  <p>The number of sprocket holes in this area must be even.</p>				
Action	Make reading possible by making bit 1 of parity V-check valid/invalid parameter TAP25 invalid.				
552	PROGRAM END NOT FOUND (, ,)	A	L	S	Blue
Cause	<p>A machining program in which EOR precedes the end M-code (M02, M30 or M99) or the next O number (work number) was loaded.</p> <p>The end-of-program condition can be changed by varying the settings of the parameter (TAP27).</p>				
Action	Since the machining program has already been loaded, the PROGRAM display must be called and then one of the above three end M-codes must be inserted in the program.				
553	WORK No. UNITS EXCEEDED (, ,)	B	L	S	Blue
Cause	An attempt has been made to load the program of a work number of more than four digits in spite of the fact that the maximum allowable number of digits in one work number is set to four.				
Action	Check bit 3 of parameter TAP26 .				
554	POWER OFF IN EIA/ISO CONVERT (, ,)	A	L	S	Blue
Cause	Power has been turned off during EIA/ISO conversion.				
Action	Check the EIA/ISO program being converted. If an anomaly is found, erase the program and repeat the conversion.				
555	— (, ,)				
Cause	—				
Action	—				
556	— (, ,)				
Cause	—				
Action	—				
557	DESIGNATED DIRECTORY NOT FOUND (, ,)	B (A)	I (L)	O (S)	Red (Blue)
Cause	The designated directory does not exist.				
Action	Check if the designated directory exists.				
	(, ,)				
Cause					
Action					

No.	Message	Type of error	Stopped status	Clearing procedure	Display
563	PRINTER I/O ERROR (Cause, ,)	G	L	S	Blue
Cause	An error in printer communications has occurred during measurement result printing.				
Action	<ul style="list-style-type: none"> Check the printer for correct connection. Check for differences in the RS-232C parameter settings between the printer and NC unit. 				
564	— (, ,)				
Cause	—				
Action	—				
565	ID MIS-CONNECTED (, ,)	G	L	S	Blue
Cause	Erroneous cable connection has occurred during connection of the ID unit and the NC unit, or power has remained turned off.				
Action	<p>Check for incorrect cable connections. Or check if the power is turned on.</p>				
566	POWER STOPPED DURING ID OPER. (, ,)	A	L	S	Blue
Cause	Power has been turned off during the operation of the ID unit.				
Action	Check the current tool data, and if errors are found, operate the unit once again.				
567	ID I/O ERROR (, ,)	G	L	S	Blue
Cause	Communication between the NC unit and the ID unit has become interrupted because of hardware trouble (such as line noise).				
Action	Contact Mazak Technical Center or Technology Center.				
568	IBS COMMUNICATION ERROR (Note 1, Note 2,)	E	I	P	Blue
Cause	An error occurred in bar loader communication with the IBS (intelligent bar loader system). Note : <ol style="list-style-type: none"> Details of communication error <ol style="list-style-type: none"> Transmission of the schedule data does not start. Writing of the material length and cut-off position does not complete. Inconsistent work number. Transmission of top-cut work number does not start. Inconsistent top-cut work number Schedule number that caused communication error. 				
Action	Press the reset key and, on the BL SCHEDULE display, press the [SCHEDULE OPERAT.] menu key again to highlight it.				
569	— (, ,)				
Cause	—				
Action	—				
570	NO DNC OPTION (, ,)	A	L	S	Blue
Cause	DNC operation was attempted although DNC option is not provided.				
Action	Provide the NC unit with a DNC option. (Only with this option, DNC operation can be carried out.)				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
571	ILLEGAL FORMAT (, ,)	A	L	S	Blue
Cause	Data other than that of the M640 Series or of the MATRIX has been received from the host (the format of the message is incorrect).				
Action	Confirm whether the received data from the host is for the M640 Series or the MATRIX.				
572	CANNOT LOAD (PROG SIZE EXCEED) (WNo., ,)	A	L	S	Blue
Cause	The contents of the transmitted machining program from the host system are not correct. (More than 2000 lines of MAZATROL program data have been transmitted.)				
Action	Check the size of the program which has been transmitted from the host system.				
573	CANNOT LOAD (TOO MANY PROGRAMS) (WNo., ,)	A	L	S	Blue
Cause	An attempt has been made to load more machining programs than the maximum number of programs that can be registered within the NC unit.				
Action	Delete unnecessary programs, or save the programs onto an external storage and then delete them. After that, load the particular program.				
574	CANNOT LOAD (AUTO OPERATION) (, ,)	A	L	S	Blue
Cause	An attempt has been made during automatic operation to load data other than machining program data.				
Action	Load such data only after completion of automatic operation.				
575	CANNOT LOAD (MISMATCH) (, ,)	A	L	S	Blue
Cause	Loading has been attempted when the transmitted data from the host system does not match to the data or other parameter settings within the NC unit (mismatching in data size, etc.).				
Action	Check if the data that has been transmitted from the host system is that which is to be used for the machine being used.				
576	SAME PROGRAM No. DESIGNATED (WNo., ,)	A	L	S	Blue
Cause	An attempt has been made to load the machining program that has the same work number as that of a machining program registered within the NC unit.				
Action	Check for an overlapping work number. This alarm message also implies that the parameter (DNC26 , bit 2) is set for the priority of the old program over a new one. If the parameter is set to 0, the old data will automatically be deleted in such a case as mentioned above and the new program data can be loaded with the specified work number.				
577	NO DESIGNATED PROGRAM (WNo., ,)	A	L	S	Blue
Cause	<ul style="list-style-type: none"> • The machining program whose transmission from the NC unit to the host system has been attempted does not exist within the NC unit. • The machining program that has been designated using a control command (work number search or program deletion) does not exist within the NC unit. 				
Action	Check if the machining program with the specified work number exists in the NC unit.				
578	MEMORY CAPACITY EXCEEDED (WNo., ,)	A	L	S	Blue
Cause	An attempt has been made to load more machining programs than the maximum number of programs that can be registered within the NC unit.				
Action	Delete unnecessary programs, or save the programs onto an external storage and then delete them. After that, load the particular program.				
579	MEMORY PROTECT (, ,)	A	L	S	Blue
Cause	Loading has been attempted when the PROGRAM LOCK/ENABLE switch setting was LOCK.				
Action	Set the switch to ENABLE, and then carry out the loading operation. This alarm message also implies that the setting of the parameter (DNC26 , bit 3) is OFF (0). Change this parameter setting to ON (1). Data loading will then become possible.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
580	CARD NOT READY (, ,)	A	L	S	Blue
Cause	A memory card has not been correctly mounted in the NC unit.				
Action	Check if the memory card is correctly mounted.				
581	DIRECTORY DOES NOT EXIST (, ,)	A	L	S	Blue
Cause	The floppy disk has not been set properly in the floppy disk drive.				
Action	Set the floppy disk properly in the floppy disk drive.				
582	DESIGNATED FILE NOT TRANSFERED (, ,)	A	L	S	Blue
Cause	A file different from the one that has been requested from NC unit to the host system was transferred from the latter.				
Action	Check the details of the file that has been transferred from the host system.				
583	PROGRAM DATA TYPE INCORRECT (, ,)	A	L	S	Blue
Cause	An attempt has been made to load a machining program that is different in structure from those stored within the NC unit.				
Action	<ul style="list-style-type: none"> Confirm that the program to be received from the host is for the M640 Series or the MATRIX. Check if the contents of the file transfer message (header block) are correct. 				
584	RECEIVED DATA TYPE INCORRECT (, ,)	A	L	S	Blue
Cause	<ul style="list-style-type: none"> An attempt has been made to load data other than machining program data and also different in structure from the data stored within the NC unit. The contents of the header block or data block in the file transfer message (including machining programs) are not correct. 				
Action	<ul style="list-style-type: none"> Check if the data that has been transferred from the host system is for use with MATRIX or for use with the machining being operated. Check the contents of the header block (version number, etc.) or data block (sequence number, etc.) in the file transfer message. 				
585	CABLE MIS-CONNECTED (, ,)	G	L	S	Blue
Cause	This message implies incorrect cable connection between the host system and the NC unit or implies a power-off status.				
Action	<ul style="list-style-type: none"> Check if the DNC cables are correctly connected. Check if the host system is turned on and ready for data transmission/reception. <p>There may be cases that although a DNC option is provided, DNC itself is not to be used for the time being and thus the DNC cables are not yet connected. If this is the case, then set the appropriate parameter (DNC26, bit 1) to OFF (0). This will clear the alarm display.</p>				
586	SYSTEM ERROR (, ,)	E	L	S	Blue
Cause	An error has occurred within the system.				
Action	Contact Mazak Technical Center or Technology Center. (At this time, also please notify them of what kind of operating procedure you had carried out before the alarm message appeared and what values were displayed in parentheses.)				
587	PROG. OPERATION NOT ALLOWED (WNo., ,)	A	L	S	Blue
Cause	<ol style="list-style-type: none"> An attempt has been made to transmit a display inhibiting program to the host system. (Program management function) An attempt has been made to transmit to the host system the program being edited (or the program being loaded using another I/O unit). 				
Action	<ol style="list-style-type: none"> Check if the specified work number is for the program of display inhibition. Carry out the transfer operation only after completion of the program editing (or program loading using another I/O unit). 				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
588	DATA OPERATION NOT ALLOWED (, ,)	A	L	S	Blue
Cause	<ul style="list-style-type: none"> An attempt has been made during automatic operation to load data other than machining program data. An attempt has been made to transmit to the host system the data being loaded using another I/O unit. An attempt has been made to load the data being saved using another I/O unit. 				
Action	Wait until automatic operation has been completed (or until the loading or saving operation using another I/O unit has been completed).				
589	DATA SIZE EXCEEDED (WNo., Note,)	A	L	S	Blue
Cause	<p>The EIA/ISO machining program includes a block that consists of more than 256 characters. (EOB or EOR is not present within 256 characters.)</p> <p>Note : The number displayed next to the work number is a line number, which corresponds to the number displayed in the lower right section of the PROGRAM display.</p>				
Action	Correct the machining program. (Insert EOB within 256 characters.)				
590	DNC COMMAND IMPOSSIBLE (, ,)	A	L	S	Blue
Cause	<p>The particular status of the NC unit disables execution of the control command that has been requested from the host system.</p> <ul style="list-style-type: none"> A request for work number search has been made during automatic operation. During automatic operation, a request for deleting the machining program being used for the automatic operation has been made. 				
Action	Wait until the NC unit becomes ready for processing or until the automatic operation is completed, and then make the request once again.				
591	NO OPTION (WNo., ,)	A	L	S	Blue
Cause	An attempt has been made to load (save) the data not supported by the current option of the DNC unit.				
Action	Only data supported by the option can be processed.				
592	ILLEGAL COMMAND RECEIVED (, ,)	A	L	S	Blue
Cause	<ul style="list-style-type: none"> The control command or file transfer command that has been requested from the host system is a nonexistent command. The machine number that has been designated for the loading of data other than machining program data does not agree with any of the machine numbers within the NC unit. 				
Action	<ul style="list-style-type: none"> Check the details of the command message that has been sent from the host system. Check if the machine number is the same as that registered within the NC unit (parameter DNC19). 				
593	DNC I/O ERROR (, ,)	G	L	S	Blue
Cause	<ul style="list-style-type: none"> During use of DNC, processing has been aborted by line noise or other hardware factors. RS-232C communication parameter settings (such as those of the baud-rate, etc.) between the host system and NC unit differ. Timer, number-of-retries or other settings are not correct. 				
Action	<ul style="list-style-type: none"> Make line checks and hardware checks of the host system and NC unit. Match the RS-232C communication parameter settings between the host system and NC unit. Set the timer, number-of-retries, or other settings to those of the host system. (Parameters for the NC unit: DNC parameters) 				
594	SEND-RECEIVE ERROR (, ,)	G	L	S	Blue
Cause	<ul style="list-style-type: none"> The preset number of retries has been exceeded during transmission/reception of command messages. RS-232C communication parameter settings (such as those of the baud-rate, etc.) between the host system and NC unit differ. Timer, number-of-retries or other settings are not correct. 				
Action	<ul style="list-style-type: none"> Make line checks and message checks of the host systems. Match the RS-232C communication parameter settings between the host system and NC unit. Set the timer, number-of-retries or other settings to those of the host system. (Parameters for the NC unit: DNC parameters) 				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
595	FILE TRANSFER ERROR (, ,)	G	L	S	Blue
Cause	<ul style="list-style-type: none"> The preset number of retries has been exceeded during transmission/reception of the messages. RS-232C communication parameter settings (such as those of the baud-rate, etc.) between the host system and NC unit differ. Timer, number-of-retries or other settings are not correct. 				
Action	<ul style="list-style-type: none"> Make line checks and message checks of the host systems. Match the RS-232C communication parameter settings between the host system and NC unit. Set the timer, number-of-retries or other settings to those of the host system. (Parameters for the NC unit: DNC parameters) 				
596	DNC MALFUNCTION (, ,)	G	L	S	Blue
Cause	An irretrievable hardware error has occurred during reception of the first message (ENQ) from the host system.				
Action	After making hardware checks of the NC and host systems and line checks, turn the NC unit power back on and then restart the receiving operation.				
597	POWER OFF DURING DNC OPERATION (, ,)	A	L	S	Blue
Cause	Power has been turned off during DNC operation.				
Action	Check for errors in the machining program being used, and if errors are found, carry out the DNC operation once again. Note, however, that if the machining program is being loaded, then loading must be carried out once again after erasing the loaded contents of the program.				
598	NO EIA/ISO OPTION (, ,)	A	L	S	Blue
Cause	An attempt has been made to transfer EIA/ISO program although the NC unit is not provided with an EIA/ISO option.				
Action	Without an EIA/ISO option, EIA/ISO program processing is not possible.				
599	DESIGNATED DATA NOT FOUND (, ,)	A	L	S	Blue
Cause	The host system has issued a request for transmission/reception of data not existing within the NC unit. <ul style="list-style-type: none"> A drum-tool data transfer request has been issued to the NC unit though it is not provided with a drum. A request for transfer of a larger volume of data than the control data stored within the NC unit has been made. 				
Action	Check the contents of the command messages that have been sent from the host system.				

3-2-7 No. 600 - No. 699, No. 1600 - No. 1699 (MAZATROL program error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display
600	— (, ,)				
Cause	—				
Action	—				
601	SYSTEM ERROR (, ,)	E	I (L)	O (S)	Red (Blue)
Cause	A processing error has occurred within the NC unit.				
Action	Using data I/O operation, save the program data, tool data, tool file data, parameters, etc. that are currently being used. After that, contact Mazak Technical Center or Technology Center.				
602	PROG. OPERATION NOT ALLOWED (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to start the program being transferred.				
Action	After the transfer operation is completed, start the program.				
603	NO DESIGNATED PROGRAM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ul style="list-style-type: none"> The program having the work number that has been set in the subprogram unit does not exist within the memory. No work number has been set in the subprogram unit. The work number that has been designated as the restart position does not exist within the memory. 				
Action	Review the machining programs to see if the designated program exists.				
604	NO PITCH IN MULTI WORKPIECES (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ul style="list-style-type: none"> Pitch X is not yet set in spite of the fact that multi-piece machining in the direction of the X-axis is to take place. Pitch Y is not yet set in spite of the fact that multi-piece machining in the direction of the Y-axis is to take place. 				
Action	Review the particular machining program and then set an appropriate multi-piece machining pitch in the common unit.				
605	NO TOOL DATA IN PROGRAM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The point-, line- or face-machining (including 3-D) unit does not contain any tool sequences.				
Action	Review the particular machining program to see if there are units that do not contain necessary tool sequences.				
606	NO FIGURE IN PROGRAM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The point-, line- or face-machining (including 3-D) unit does not have any shape data.				
Action	Review the particular machining program to see if there are units that do not contain necessary shape data.				
607	MISSING INPUT DATA (POINT) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A point-machining unit lacks data.				
Action	Review the particular machining program, and set data if a point-machining unit lacks data.				
608	MISSING INPUT DATA (LINE) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A line-machining unit lacks data.				
Action	Review the particular machining program, and set data if a line-machining unit lacks data.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
609	MISSING INPUT DATA (FACE) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A face-machining unit lacks data.				
Action	Review the particular machining program, and set data if a face-machining unit lacks data.				
610	MISSING TOOL DATA FOR POINT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A point-machining tool sequence lacks data.				
Action	Review the particular machining program, and set data if a point-machining tool sequence lacks data.				
611	MISSING TOOL DATA FOR LINE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A line-machining tool sequence lacks data.				
Action	Review the particular machining program, and set data if a line-machining tool sequence lacks data.				
612	MISSING TOOL DATA FOR FACE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A face-machining (including 3-D) tool sequence lacks data.				
Action	Review the particular machining program, and set data if a face-machining tool sequence lacks data.				
613	DATA MISSING IN WPC UNIT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The WPC unit lacks data.				
Action	Review the particular machining program, and set data if the WPC unit lacks data.				
614	SUB PROGRAM NESTING EXCEEDED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The maximum permissible number of repeats of MAZATROL program nesting has been exceeded nine.				
Action	Review and correct the particular machining program so that the total number of repeats of nesting does not exceed nine.				
615	DATA MISSING IN OFFSET UNIT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The offset unit lacks data.				
Action	Review the particular machining program, and set data if the offset unit lacks data.				
616	DATA ERROR IN M CODE UNIT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The M-code unit contains no data.				
Action	Review the particular machining program, and input data to the M-code unit.				
617	EXECUTION IMPOSSIBLE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The data processing operation cannot be performed because of contradiction in data. This condition occurs if an attempt is made to start automatic operation when the specified work number is an unregistered number.				
Action	Search out the contradictory data making reference to WNo., UNo., SNo. (which are displayed together with the alarm message), and then correct the data.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
618	POINT CUTTING PARAMETER ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The point-machining parameter setting(s) is out of its permissible range.				
Action	——				
619	LINE/FACE CUTTING PAR. ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The line- or face-machining parameter settings are out of their permissible ranges.				
Action	The parameter E13 is set to "0"; change it to a value other than "0".				
620	CUTTING SPEED ZERO (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Of tool sequence data (except for chip removal), the surface speed (C-SP) is unset or set to "0".				
Action	Review the machining program and set the desired surface speed (C-SP).				
621	FEEDRATE ZERO (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Of tool sequence data (except for chip removal), the feedrate (FR) is unset or set to "0".				
Action	Review the machining program and set the desired feedrate (FR).				
622	DESIGNATED UNIT NOT FOUND (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	The unit that has been designated as the restart position is not present in the program with the specified work number.				
Action	Review the machining program and designate the correct unit number.				
623	DESIGNATED SNo. NOT FOUND (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	The tool sequence that has been designated as the restart position is not present in the unit of the specified work number; two or more lines of tool sequence data are present in the line-machining chamfering unit.				
Action	Review the machining program and designate the correct tool sequence number.				
624	RESTART IMPOSSIBLE (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	<ul style="list-style-type: none"> • The unit that has been designated as the restart position is the end unit. • The designated number of times of reappearance (L) is too large and the corresponding restart position is not present. • The restart data is incomplete. 				
Action	Check the contents of the restart data or the program.				
625	ENDMILL DIAMETER EXCEEDED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ul style="list-style-type: none"> • The value of "(groove width) – (finish allowance R) × 2" of the endmill groove unit is smaller than the "tool diameter" value of the rough-machining tool. • The "groove width" value of the endmill groove unit is smaller than the "tool diameter" value of the finishing tool. 				
Action	——				
626	DESIGNATED TOOL NOT FOUND (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The tool(s) specified in the program is not registered in the tool data.				
Action	Carry out a tool layout operation and register the necessary tool(s) on the TOOL DATA display.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
627	TOOL DATA INPUT PROCESS ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The data of the tool length or tool diameter is not yet input on the TOOL DATA display.				
Action	Review the tool data and input the tool length or tool diameter.				
628	NO ASSIGNED TOOL IN TOOL FILE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The tool specified in the program is not registered on the TOOL FILE display.				
Action	Register the tool data that is to be used in the program into the tool file.				
629	TOOL FILE INPUT PROCESS ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The tool file lacks of data.				
Action	Review the data on the TOOL FILE display and fill in any empty items with data.				
630	Z DEPTH OF CUT TOO LARGE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Of the line- or face-machining tool sequence data, the value of the Z depth of cut is in excess of the depth of cut on the TOOL FILE display.				
Action	Review the machining program and correct the value of the Z depth of cut.				
631	STOCK REMOVAL R TOO LARGE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The value of "(removal allowance R) – (finish allowance R)" in the line-machining unit is larger than the value of the tool diameter of the rough-cutting tool. The value of removal allowance R in the line-machining unit is larger than the value of the tool diameter of the finishing tool.				
Action	Review the machining program and correct the values of removal allowance R and finishing allowance R in the line-machining unit.				
632	RADIAL DEPTH OF CUT ZERO (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Of line- or face-machining tool sequence data, the radial depth of cut is set to zero or smaller.				
Action	Review the machining program and set the radial depth of cut to the correct value.				
633	Z DEPTH OF CUT ZERO (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Of line- or face-machining tool sequence data, the Z depth of cut is set to zero or smaller.				
Action	Review the machining program and set the Z depth of cut to the correct value.				
634	FINISH DEPTH OF CUT ZERO (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The finish allowance value in the line- or face-machining unit is set to zero in spite of the fact that a finishing tool is registered.				
Action	Review the machining program and set data in the finish allowance data item.				
635	TOOL DIAMETER ZERO (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Of tool data, the tool diameter setting is zero.				
Action	Review the data on the TOOL DATA display and set data in the tool diameter item.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
636	STOCK REMOVAL Z TOO SMALL (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the line- or face-machining unit, removal allowance Z is smaller than finish allowance Z.				
Action	Review the line- or face-machining unit and increase removal allowance Z to a value greater than that of finish allowance Z.				
637	STOCK REMOVAL R TOO SMALL (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the line- or face-machining unit, removal allowance R is smaller than finish allowance R.				
Action	Review the line- or face-machining unit and increase the value of removal allowance R to a value greater than that of finish allowance R.				
638	R DEPTH OF CUT TOO LARGE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Of the face-machining tool sequence data, the setting of the radial depth of cut is smaller than the tool diameter setting on the TOOL DATA display.				
Action	Review the machining program and increase the radial depth of cut to a value greater than the tool diameter setting in the tool data.				
639	DESIGNATED PALLET NOT FOUND (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The pallet number that has been set in the pallet changing unit is larger than the maximum allowable number of pallets set in the parameter L46 .				
Action	Review the machining program and set an allowable pallet number.				
640	ILLEGAL INDEX ANGLE INPUT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The data that has been set in the angle item of the indexing unit is that which cannot be divided by the parameter L37 setting (minimum allowable angle of index).				
Action	Review the machining program and set an allowable angle of index.				
641	MISSING INPUT DATA (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The pallet changing unit or the indexing unit lacks of unit data. Initial point Z is not yet set in the common unit. CPT-X, CPT-Z or rotational direction is not yet set in the mill-turning unit. When "SHIFT - R.T" is set in the WPC shift unit with the machine equipped with table selection function, the rotational center is not set in the table selection unit. The table is not set in the table selection unit.				
Action	Review the machining program and set all the necessary values in the unit.				
642	ILLEGAL NEXT PALLET No. INPUT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The same pallet number as the current pallet number has been set as the next one.				
Action	Review the machining program and make sure of the pallet numbers.				
643	DATA ERROR IN MAN. PROG. UNIT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The manual program mode unit contains a sequence that has no data.				
Action	Review the machining program, and fill in any incomplete sequence with data or erase such sequences.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
644	NOMINAL DIAMETER NOT FOUND (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The nominal diameter item of the tool sequence data is not complete. The nominal diameter item of the MMS unit or the manual program mode unit (when a tool is set) is not complete.				
Action	Review the machining program, and set data in the nominal diameter item of the MMS unit or the manual program mode unit (when a tool is set) or erase the corresponding portion.				
645	PRIORITY No. OVERLAP (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The same priority number is assigned to different tools.				
Action	Within one process, the same priority number must not be assigned to different tools. Change the priority number.				
646	ILLEGAL PRIORITY NUMBER (WNo., UNo., SNo.)* (LNo. 1, LNo. 2,)**	B	I (L)	O (S)	Red (Blue)
Cause	The priority numbering order within a unit is not correct. * During setting the priority numbers on the PROGRAM display. ** During setting the priority numbers on the PROGRAM LAYOUT display - data in the layout lines LNo. 1 and LNo. 2 are not correct.				
Action	The machining order within one unit has been reversed by the incorrect priority numbering. Change the priority numbers.				
647	END UNIT NOT FOUND (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The end unit is not present in the program.				
Action	Review the machining program and set the end unit at the end of the program.				
648	MULTI OFFSET DATA TOO LARGE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	More than 10 sets of offset data have been input for multi-piece machining.				
Action	The machining program is in an abnormal state. If the program is already saved onto cassette tape, floppy disk or other media, erase the program and then reload it. If the program is not yet saved, make corrections with the editing function and fully scan for more data errors.				
649	MEASURING SEQUENCE INCOMPLETE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The measurement sequence lacks of data.				
Action	Review the machining program, and input data to the measurement sequence to make it complete.				
650	CHAMFERING IMPOSSIBLE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Cutting is impossible because the chamfering cutter is likely to come into contact with the wall or bottom of the workpiece during chamfering. The data of the specified chamfering cutter on the TOOL DATA or TOOL FILE display is not appropriate.				
Action	Review the machining program or the tool file, and correct inappropriate data.				
651	GEAR PARAMETER ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to execute the point-, line- or face-machining MAZATROL program when the setting of parameter SA51 was "5" or more.				
Action	Change the setting of parameter SA51 to a value between 0 and 4.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
652	GEAR SHIFT DATA ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The tool sequence data contains an unavailable milling spindle gear-shift M-code(s).				
Action	Change the corresponding code(s) to an available one(s). 2-gear $\begin{cases} H : M39 \\ L : M38 \end{cases}$ 3-gear $\begin{cases} H : M39 \\ M : M38 \\ L : M37 \end{cases}$ 4-gear $\begin{cases} H : M39 \\ MH : M38 \\ ML : M37 \\ L : M36 \end{cases}$				
653	ILLEGAL TOOL DESIGNATED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Tools that cannot be used have been designated.				
Action	Review the machining program and designate tools that are usable.				
654	TOOL DATA ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The tool length and tool diameter settings on the TOOL DATA display are negative.				
Action	Set positive tool length and tool diameter values.				
655	PROGRAM DATA CORRUPT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The program is destroyed.				
Action	Erase a part of the program and then re-create the destroyed part; or erase the entire program and then carry out a loading operation using the data I/O operation once again.				
656	MASURING SEQ. DATA NOT FOUND (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The measurement units include one that has no sequence data.				
Action	Create one or more lines of sequence data in the corresponding measurement unit, or erase the unit.				
657	ILLEGAL NUMBER INPUT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The program contains incorrect data.				
Action	Review the machining program and make data corrections.				
658	INITIAL Z < MATERIAL DEPTH (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The setting of the material height in the 3-D machining unit is greater than that of initial point Z in the common unit.				
Action	Change the program to give a material height value smaller than the initial point Z value.				
659	NO TOOL PATH CHECK (I/O BUSY) (WNo., ,)	A	L	S	Blue
Cause	The tool path check cannot be performed since I/O operation (loading) is in progress.				
Action	Make the tool path check after the I/O operation has been completed.				
660	CANNOT MOVE DESIGNATED AXIS (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	The Y-axis or Z-axis of the index position has been appointed using the indexing unit when the parameter L41 is set to "2".				
Action	Using the data cancellation key, erase the Y-axis or Z-axis data of the index position.				

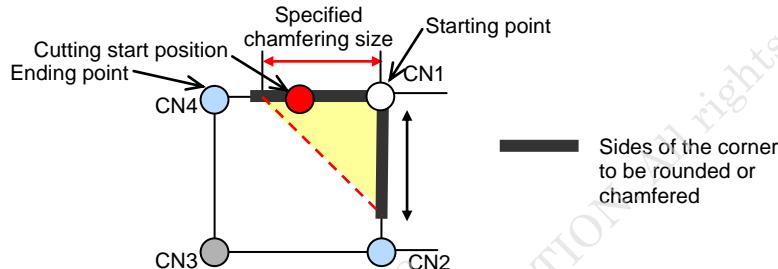
No.	Message (WNo., UNo., SNo.)	Type of error	Stopped status	Clearing procedure	Display
661	ILLEGAL M CODE (WNo., UNo., SNo.)				
Cause	M195 (tool breakage detection start command code) has been set for the M-code unit or for the manual program mode sequence.				
Action	M195 cannot be executed on MAZATROL programs. Delete that command code from the program.				
662	NO INCLINED PLANE OPTION (WNo., UNo.,)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to execute the inclined-plane machining program in the absence of an inclined-plane machining option.				
Action	Inclined-plane machining is not possible because of the absence of an inclined-plane machining option.				
663	WRONG HEAD ANGLE (WNo., UNo.,)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> 1. A corner-face unit or plane inclination measurement has been designated in unit data other than inclined-plane machining unit data. 2. Table rotational machining has been designated in spite of the fact that the facial angle data in the face definition sequence is not for the top plane. 3. Calibration measurement has been designated for the top plane or an inclined plane. 4. Groove center, hole center, boss center, and step width measurements have been designated for an inclined plane. 5. Machining at the work hand side “↓Z” (B-axis at 90°, approach in the Z-axis) has been specified for the direction of the geometry sequence in coordinate measurement. 6. Machining at the work hand side “↓Z” (B-axis at 90°, approach in the Z-axis) has been specified for the direction of the geometry sequence in workpiece measurement. 7. Measurement at the work hand side has been attempted with the parameter setting of L106 bit5 = 0 (B-axis user-defined angle measurement invalid). 				
Action	<ol style="list-style-type: none"> 1. A corner-face unit and plane inclination measurement can be designated only for an inclined plane. 2. Table rotational machining can be executed only for the top plane. 3. Calibration measurement is possible only for the side. 4. Only reference plane measurement and plane inclination measurement are possible for inclined planes. 5. Change the direction setting of the geometry sequence in coordinate measurement to “↓X” (B-axis at 90°, approach in the X-axis). Alternatively, select “↓X”, “←Z”, or “←X” for measurement with the B-axis at 0°. 6. Change the direction setting of the geometry sequence in workpiece measurement to “↓X” (B-axis at 90°, approach in the X-axis). Alternatively, select “↓X”, “←Z”, or “←X” for measurement with the B-axis at 0°. 7. Change the parameter setting to L106 bit5 = 1. 				
664	3-D UNIT NOT FOUND IN PROGRAM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A 3-D machining unit has been set in the program in spite of the fact that 3-D machining option is not provided.				
Action	Erase the 3-D machining unit from the machining program.				
665	ILLEGAL DATA IN 3-D UNIT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The 3-D machining unit lacks of unit data.				
Action	Review the machining program and set necessary data in the 3-D machining unit.				
666	PLANE DATA NOT FOUND IN PROGRAM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The 3-D machining unit lacks of plane definition data.				
Action	Review the machining program and set plane definition data in the 3-D machining unit.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
667	CHECK SURFACE DATA NOT FOUND (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The 3-D machining unit lacks of check surface data.				
Action	Review the machining program and set check surface data in the 3-D machining unit.				
668	ILLEGAL PLANE DATA IN PROGRAM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The plane definition data in the 3-D machining unit is not complete.				
Action	Review the machining program and set data in the plane definition data item.				
669	ILLEGAL TOLERANCE DATA INPUT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The value of the tolerance parameter that has been designated in the tool sequence is "0".				
Action	Check the parameters E67 through E75, and set a value other than "0" in the parameter whose setting is "0".				
670	ILLEGAL SEQUENCE DATA IN PRG. (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The tool sequence in the 3-D machining unit lacks of data.				
Action	Review the machining program and input data to the tool sequence.				
671	ILLEGAL MOVE SURFACE DATA (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The coordinate conversion data in the 3-D machining unit is not complete.				
Action	Review the machining program and make the coordinate conversion data complete.				
672	ILLEGAL AREA DATA INPUT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the check surface data of the 3-D machining unit, the setting of the maximum value is smaller than that of the minimum value.				
Action	Review the check surface data, and make corrections so that the setting of the maximum value is equal to or greater than that of the minimum value for each axis.				
673	FL NUMBER EXCEEDED (3-D UNIT) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the ruled-surface unit, the number of FLs is in excess of 20, or in the line- or face-machining unit, the number of defined figures is in excess of 2.				
Action	Review the machining program and correct the shape data.				
674	NO 5FACE CUTTING OPTION (WNo., UNo.,)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to execute a five-surface machining program when the five-surface machining option was not present.				
Action	Set the five-surface machining option to execute a five-surface machining program.				
675	ILLEGAL CUTTING FACE DESIGNATED (WNo., UNo., SNo.)	B	I (L)	P (S)	Red (Blue)
Cause	A face that cannot be cut with the selected head has been designated.				
Action	Change either the selected face or the head.				

No.	Type of error	Stopped status	Clearing procedure	Display	
676	ILLEGAL UNIT (WNo., UNo.,)	B	I (L)	O (S)	Red (Blue)
Cause	<p>1. The face definition unit or five-surface machining unit includes illegal units and measurement sequences. Illegal units and sequences: face definition unit, pallet change unit, process end unit, index unit, WPC unit, and measurement of workpiece inclination.</p> <p>2. An MMS unit has been set in the program that has table rotational machining designated in its face definition sequence.</p> <p>3. Executing the mill-turning unit has been attempted in a machine not having a mill-turning option.</p> <p>4. An attempt was made to run a program including a table selection unit on the machine not equipped with table selection function or the machine equipped with table selection function but while the function is invalid (parameter D106 bit 4 = 0).</p> <p>5. C-axis machining has been selected for machining at the work hand side. Or, turning has been selected for machining at the work hand side.</p> <p>6. Workpiece measurement has been executed with the work hand at 0° other than in the workpiece transfer unit.</p>				
Action	<p>1. Remove all illegal units and sequences from the machining program.</p> <p>2. MMS measurement is not possible while table rotational machining is effective.</p> <p>3. The mill-turning unit cannot be executed without a mill-turning option.</p> <p>4. With the machine not equipped with table selection function, correct the program into a program without table selection unit. With the machine equipped with table selection function, make table selection valid (parameter D106 bit 4 = 1) and check the program and tool path before starting machining.</p> <p>5. Delete the C-axis machining unit and turning unit at the work hand side.</p> <p>6. Set the work hand angle to 0° in the workpiece transfer unit.</p>				
677	UNREGISTERED HEAD DATA (WNo., UNo.,)	B	I (L)	P (S)	Red (Blue)
Cause	Head data corresponding to the head of the designated head number does not exist.				
Action	Review the designated head number.				
678	NO INTERSECTION (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the line- or face-machining unit, the coordinates of the intersection point of figures cannot be obtained because of shortage of, or contradiction, in the free-shape data.				
Action	—				
679	CONNECTING CORNER IMPOSSIBLE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The figures cannot be connected smoothly at corner R because of contradiction in the data of corner R or in the data of the figures in front and rear of corner R.				
Action	Review the machining program and check the value of corner R of the free shapes.				
680	NUMBER OF HOLES EXCEEDED (>500) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The point-machining units include one(s) that has more than 500 holes defined in it.				
Action	Review the point-machining units, and make corrections so that the total number of hole settings in one point-machining unit is not greater than 500.				
681	CORNER R/C DEFINED AT SPT/FPT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Corner rounding or corner chamfering has been set at the starting or ending point of a figure when defining figures in the central linear machining, right-hand linear machining, left-hand linear machining, right-hand chamfering or left-hand chamfering units.				
Action	Review the machining program and correct it so that the corner rounding or corner chamfering is not set at the starting or ending point.				

No.	Message (WNo., UNo., SNo.)	Type of error	Stopped status	Clearing procedure	Display
682	ILLEGAL REPEAT FIGURE Contradiction presides in the figure rotation or figure shift data that has been set during defining free figures in the line- or face-machining unit.	B	I (L)	O (S)	Red (Blue)
Action	Review and correct the corresponding shape data.				
683	UNDEFINED CORNER The specified size of corner rounding or chamfering is inadmissible. Example : This alarm does not occur in general unless the specified size of rounding or chamfering exceeds the length of one of the two sides of the corner concerned. As for a rectangular line- or face-machining shape described with the aid of the [SQUARE] menu function, in particular, the size in question for corners CN1 (starting point) and CN4 (ending point) must not exceed half the length of the side between them since the cutting start position will be set in actual machining to the mid-point of that side.	B	I (L)	O (S)	Red (Blue)
Cause					
Action	Check the shape sequence data in question to correct the size of corner rounding or chamfering. In the above-mentioned particular case of rectangular shape, use the [ARBITRARY] menu function, instead of [SQUARE], to describe the desired shape on multiple lines of shape sequence data.				
684	POINT CUTTING PATTERN ERROR The point-machining shape definition data is not appropriate.	B	I (L)	O (S)	Red (Blue)
Action	Review and correct the corresponding shape data.				
685	SQUARE CANNOT BE DEFINED When the shape pattern is "square", the input data cannot be used to make shape definitions.	B	I (L)	O (S)	Red (Blue)
Action	Review and correct the corresponding shape data.				
686	NO STARTING POINT During input of free-shape data (open-figure data) to the line-machining unit, "?" has been set as the definition of the starting point.	B	I (L)	O (S)	Red (Blue)
Action	Review the machining program and set the coordinates of the starting point of the free shape.				
687	NO FINISH POINT During input of free-shape data (open-figure data) to the line-machining unit, "?" has been set as the definition of the ending point.	B	I (L)	O (S)	Red (Blue)
Action	Review the machining program and set the coordinates of the ending point of the free shape.				
688	INSUFFICIENT INPUT DATA The coordinates of the intersection point cannot be calculated since the free-shape input data in the line- or face-machining unit is incomplete.	B	I (L)	O (S)	Red (Blue)
Action	Review the corresponding shape data and set data that is wanting.				

No.	Message (WNo., UNo., SNo.)	Type of error	Stopped status	Clearing procedure	Display
689	INPUT DATA TOO MANY The line- or face-machining unit contains too much free-shape input data, and there is contradiction between overlapping data. Too many tool sequences have been set for the line- or face-machining unit.	B	I (L)	O (S)	Red (Blue)
Cause	1. Review the corresponding shape data and erase either one of the overlapping data sets. 2. Reduce the number of tool sequences.				
Action	1. Review the corresponding shape data and set correct data.				
690	ILLEGAL RADIUS Contradiction exists in the free-shape data that have been set to define arc in the line- or face-machining units.	B	I (L)	O (S)	Red (Blue)
Cause	Review the corresponding shape data and set correct data.				
Action	Review the machining program, and define the second shape in the endmilling-mountain (STEP), pocket milling-mountain or pocket milling-valley unit.				
691	MOUNT (VALLEY) SHAPE ERROR The second figures (inside figures) are not yet defined in the endmilling-mountain (STEP), pocket milling-mountain or pocket milling-valley unit.	B	I (L)	O (S)	Red (Blue)
Cause	Review the machining program, and define the second shape in the endmilling-mountain (STEP), pocket milling-mountain or pocket milling-valley unit.				
692	MAX POINT NUMBER EXCEEDED (>200) The number of points which are necessary to define the shapes designated in the line- or face-machining unit exceeds 200.	B	I (L)	O (S)	Red (Blue)
Cause	Review the machining program, and reduce the number of shapes within one line- or face-machining unit.				
Action	Review the machining program, and reduce the number of shapes within one line- or face-machining unit.				
693	NUMBER OF SHAPES TOO LARGE Among the line- or face-machining units is one(s) that contains more shapes than allowable within one unit.	B	I (L)	O (S)	Red (Blue)
Cause	Review the corresponding shape data and check the number of shapes.				
Action	Review the machining program and check the free-shaped data.				
694	FIXED FIGURE DESIGNATED ERROR Fixed shapes are included in the shape data (open figures) of the central linear machining, right-hand linear machining, left-hand linear machining, right-hand chamfering, left-hand chamfering or endmilling-groove units.	B	I (L)	O (S)	Red (Blue)
Cause	Change the fixed shapes to free ones.				
Action	It is not possible to draw a tangent line from point P1 since it is inside the arc.				
695	POINT INSIDE CIRCLE Review the machining program and check the free-shaped data.	B	I (L)	O (S)	Red (Blue)
Cause	Review the machining program and check the value of the crossing point (P).				
Action	"Left" or "right" is set for the crossing point (P), though "up" or "down" should have been set.				
696	ILLEGAL DIRECTION The crossing point (P) is not yet input in spite of the fact that there are more than two points of intersection with the arc.	B	I (L)	O (S)	Red (Blue)
Cause	Review the machining program and set the crossing point (P).				
Action	Review the machining program and set the crossing point (P).				
697	DATUM <P> REQUIRED The crossing point (P) is not yet input in spite of the fact that there are more than two points of intersection with the arc.	B	I (L)	O (S)	Red (Blue)
Cause	Review the machining program and set the crossing point (P).				
Action	Review the machining program and set the crossing point (P).				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
698	TWO POINTS OVERLAP (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The coordinate values of the start point and end point are the same.				
Action	For the pattern of straight line, the data of X/Y are set to exactly the same end point coordinate values as X/Y present on the preceding line of the program; delete these data.				
699	PARALLEL LINE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The two straight lines are parallel to each other, and thus the coordinates of their intersection point cannot be obtained.				
Action	Review the corresponding shape data and set correct data.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1601	ILLEGAL ATC MODE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	ATC mode ID number 5 has been set in the main program.				
Action	ATC mode ID number 5 can be set only in subprograms. Set ATC mode ID number 0 to 4 in the main program.				
1602	SET INDEX UNIT BEFORE MACHINING (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The position of W-axis is not specified in the index unit preceding the machining unit.				
Action	Review the machining program and set the position of the W-axis in the index unit.				
1603	PROGRAM IS NOT WPC. (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The main MAZATROL program of the initial-point scheme has been executed under the setting of the machine model with MAZATROL-based control of the lower turret.				
Action	Re-create the main MAZATROL program into a program of the workpiece scheme.				
1604	ILLEGAL TOOL DIRECTION (INDEX B) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	There is a mismatch in workpiece scheme data settings between the machining direction for index unit POS-B and the machining direction for the machining section of the lower-turret milling tool.				
Action	Insert an index unit in front of the desired milling unit. Next, set the value to POS-B appropriate for the lower-turret milling tool machining section, or select the appropriate lower-turret milling tool according to the particular machining direction for POS-B of the index unit.				
1605	LOW TUR CANNOT EXECUTE MIRROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Either lower-turret turning unit, the measuring unit, the manual program mode unit, or subprogram EIA call has been executed with MIRROR of the WPC shift unit made valid in a program of the workpiece scheme.				
Action	Insert the WPC shift unit in front of the desired unit and then make MIRROR invalid.				
1606	ILLEGAL CUTTING ANGLE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The infeed angle or relief angle during cutting is 0° or less.				
Action	Confirm the infeed angle or relief angle in the tool data. Change the B-axis machining angle.				
1607	NEED TOOL ROTATION DIRECTION (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	The M-SP ROT. item (which is indispensable for executing a MAZATROL program when K102 bit 6 = 1) remains unset on the TOOL DATA display.				
Action	Perform the required setting for the M-SP ROT. item.				
1608	NEED SELECT TABLE UNIT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	When table selection is valid (parameter D106 bit 4 = 1), no table selection unit is set before the WPC, WPC shift or indexing unit.				
Action	Set a table selection unit before the WPC, WPC shift or indexing unit.				
1609	CANNOT SET WPC IN 2WORKPIECE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A basic coordinates system unit (WPC) is set in the program section for the two-workpiece machining sequence.				
Action	Delete the basic coordinates system unit.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1610	SET INDEX UNIT IN 2WORKPIECE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Machining units in the two-workpiece machining section are not preceded by an index unit.				
Action	Set an index unit.				
1611	SET INDEX UNIT AFTER 2WORKPIECE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Machining units after the two-workpiece machining section are not preceded by an index unit.				
Action	Set an index unit.				
1612	SET HEAD UNIT AFTER 2WORKPIECE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Machining units after the two-workpiece machining section are not preceded by a head selection unit.				
Action	Set a head selection unit.				
1613	SET HEAD UNIT IN 2WORKPIECE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Machining units in the two-workpiece machining section are not preceded by a head selection unit.				
Action	Set a head selection unit.				
1614	NO TABLE ROTATION AXIS (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Index unit is set while NO USE is selected for ROT. AXIS in table selection unit on the machine not equipped with spindle head of swiveling type (parameter BA48 is set to 0).				
Action	Delete the index unit.				
	(, ,)				
Cause					
Action					
1616	CYLINDER CHAMFERING N/A (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt was made to execute the smooth chamfering program although the smooth chamfering function is not provided.				
Action	The smooth chamfering program cannot be executed because the smooth chamfering function is not provided.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1617	CYLINDER CHAMFERING IMPOSSIBLE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<p>1. An attempt was made to execute the smooth chamfering program with the machine type with which the smooth chamfering function is not available.</p> <p>2. An attempt was made to execute the smooth chamfering program when the B-axis angle is other than +90° in the end milling slot unit of WPC workpiece scheme.</p> <p>3. An attempt was made to execute the smooth chamfering program when the machining face is other than cylindrical surface in the C-axis point machining unit of WPC workpiece scheme.</p> <p>4. An attempt was made to execute the smooth chamfering program when HOLE-DEP is set to a value other than 0 in the chamfering tool sequence for point machining.</p> <p>5. An attempt was made to execute the smooth chamfering program when the cylinder radius is set to 0 or blank in the end milling slot unit of WPC workpiece scheme.</p> <p>6. An attempt was made to execute the smooth chamfering program when CHMF is set to 0 or blank in the end milling slot unit or point machining unit.</p>				
Action	<p>1. Delete the smooth chamfering program.</p> <p>2. Set an index unit before the machining unit that includes smooth chamfering and set angle B to +90°.</p> <p>3. Set the machining face to the cylindrical surface in the C-axis point machining unit.</p> <p>4. Set HOLE-DEP in the chamfering tool sequence to 0.</p> <p>5. Input the correct cylinder radius value in the end milling slot unit.</p> <p>6. Input the correct value for CHMF.</p>				
	(, ,)				
Cause					
Action					

3-2-8 No. 700 - No. 799, No. 1700 - No. 1799 (MAZATROL program error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display	
700	— (, ,)					
Cause	<hr/>					
Action	<hr/>					
701	DEFINED SHAPE TOO SMALL (WNo., UNo., SNo.)	B	K	O	Blue	
Cause	The shape compensation clearance with respect to the shape of the endmilling-top is too large; or the tool diameter with respect to the size of the line-inside machining is too large.	<hr/>				
Action	Change the shape compensation clearance (parameter E13) to an appropriate value; or use a tool of smaller diameter.	<hr/>				
702	FIGURE DEFINITION ERROR (WNo., UNo., SNo.)	B	K	O	Blue	
Cause	1. The input shape is contradictory to logic, that is, the radius of the arc, for example, does not agree with the distance from the center. 2. An attempt was made to execute the smooth chamfering program when CHMF in the end milling slot unit is set to other than 0, when an arc is specified or when the shape sequence is specified in three or more lines.	<hr/>				
Action	1. Such contradiction usually results from arithmetic errors. Change the radial depth of cut by some micro, or use a tool of smaller diameter. 2. When executing the smooth chamfering program, set a linear shape, whose angle formed with the X-axis is 0 degrees, in two lines (start point line and end point line).	<hr/>				
703	PROCESS DEFINITION ERROR (WNo., UNo., SNo.)	B	K	O	Blue	
Cause	The machining conditions are incorrect (for example, the radial depth of cut is zero).	<hr/>				
Action	Change the machining conditions to correct ones.	<hr/>				
704	TOOL INTERFERENCE (WNo., UNo., SNo.)	B	K	O	Blue	
Cause	In area machining, the tool diameter with respect to the figure is too large.	<hr/>				
Action	Replace the tool with one that has a smaller diameter; or select the M2 mode endmilling-mountain (STEP) machining pattern with setting bit 7 of parameter E91 if this error occurs in the outside machining endmilling-mountain (STEP).	<hr/>				
705	APPROACH POINT ERROR (WNo., UNo., SNo.)	B	K	O	Red	
Cause	1. The approach point cannot be obtained. 2. The approach path or retract path may interfere with the programmed shape.	<hr/>				
Action	1. Reduce the tool diameter, the approach amount (parameters E1, E2) and/or the overlap amount (parameter E21). 2. Extend the shape toward the approach or escape side.	<hr/>				
706	ILLEGAL FIGURE DATA (WNo., UNo., SNo.)	B	L	O	Blue	
Cause	<ul style="list-style-type: none"> • The shape has been separated into three segments or more as a result of offsetting. • The inside form does not contain the center of the outside form for outside-related fixed path. • The outside form is concave for inside-related fixed path. • The inside form is not adequate since it contains a concave or an intersection. 	<hr/>				
Action	Change the machining pattern (from inversed type to fixed type, for example); or divide the machining shape in advance so that it will not be separated by offsetting.	<hr/>				
707	CHAMFER CUTTER INTERFERENCE (WNo., UNo., SNo.)	B	K	O	Blue	
Cause	1. The chamfering tool interferes with the side wall or bottom. 2. The tool edge is not in perfect contact with the surface to chamfer.	<hr/>				
Action	1. Use a tool which does not interfere with the side wall or bottom. 2. Shorten the chamfering width or use a tool having a long cutting edge.	<hr/>				

No.	Message (WNo., UNo., SNo.)	Type of error	Stopped status	Clearing procedure	Display
708	BLOCK DATA LIMIT EXCEEDED In the EIA program, the total number of characters within one block is in excess of 248.	B	L	S	Blue
Cause	Divide the block so that one block contains 248 characters or less.				
Action					
709	CHECK DEPTH-R No automatic calculations of intersection were obtained since there is no or too short a distance between end points of the shape.	B	L	S	Blue
Cause	Change the radial depth of cut in the tool sequence by some micro.				
Action					
710	CORNER ROUNDING CUTTER DIA ERR The programmed amount of round chamfering differs from the following: $(\text{NOM-}\phi - \text{MIN-}\phi)/2$. where NOM- ϕ and MIN- ϕ are the setting values of the selected tool on the TOOL FILE display.	B	L	S	Blue
Cause	Select a tool appropriate to the chamfering amount.				
Action					
711	UNIT DATA NOT FOUND Unset unit data exists for the unit you have made an attempt to execute.	B	I (L)	O (S)	Red (Blue)
Cause	Set all data.				
Action					
712	ILLEGAL UNIT DATA Sequence data for the unit you have made an attempt to execute does not match to the unit data.	B	I (L)	O (S)	Red (Blue)
Cause	Delete the sequence data, and then set correct sequence data.				
Action					
713	SEQUENCE DATA NOT FOUND No sequence data exists for the unit you have made an attempt to execute.	B	I (L)	O (S)	Red (Blue)
Cause	Set sequence data.				
Action					
714	ILLEGAL SEQUENCE DATA Data out of the allowable setting range exists in the sequence data of the unit you have made an attempt to execute.	B	I (L)	O (S)	Red (Blue)
Cause	Correct the sequence data.				
Action					
715	ILLEGAL CUTTING POINT The specified infeed point is outside the profile of the workpiece (approximate workpiece shape designated by common unit).	B	I (L)	O (S)	Red (Blue)
Cause	Move the infeed point into the profile of the workpiece.				
Action					
716	ILLEGAL CUTTING START POINT The relative position of the starting or ending point specified by the sequence data with respect to the specified infeed point is not appropriate.	B	I (L)	O (S)	Red (Blue)
Cause	Change the position of the starting or ending point of the sequence data, or change the position of the infeed point.				
Action					

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
717	SHAPE EXCEEDS MATERIAL SIZE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The starting or ending point is outside the profile of the workpiece (approximate workpiece shape specified by common unit).				
Action	Change the starting or ending point.				
718	CUTTING DIRECTION NOT DEFINED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The direction of machining (forward/backward) cannot be determined using the specified infeed point and sequence data.				
Action	Change the position of the starting or ending point.				
719	REVERSE SHAPE CONTOUR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Part of the defined shape is opposite in direction of machining with respect to the reference axis movement direction.				
Action	Correct the position of the starting or ending point.				
720	DOUBLE SHAPE CONTOUR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The defined workpiece shape has overlaps.				
Action	Correct the position of the starting or ending point.				
721	ILLEGAL RADIUS (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Arc-drawing data you have specified is illegal. That is, the relationship between the starting point, ending point, and radius of the arc is incorrect. It is impossible to define an arc.				
Action	Correct the starting-point data, ending-point data, or radius data of the arc.				
722	LLEGAL CORNER DEFINITION (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	There are the following errors related to the designated corner C data: <BAR, CPY, CORNER> The arc length of the block present before or after corner C is smaller than that of corner C. <T, GROOV> Added corner C is outside the workpiece profile.				
Action	Correct either the length of corner C or the sequence data set before or after corner C.				
723	EXCEEDS NUMBER OF SHAPES (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Defined shape of machining or of the workpiece is too complicated to be processed.				
Action	Simplify the shape of the machining or of the workpiece.				
724	EXCEEDS NUMBER OF VALLEY SHAPES (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The total number of valleys in the machining shape is in excess of 16.				
Action	Divide the machining shape in multiple units.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
725	ILLEGAL COMMON DATA (RADIAL) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	OD-MAX and ID-MIN in the common unit do not satisfy the following condition: OD-MAX > ID-MIN ≥ 0				
Action	Set correct data to OD-MAX , or ID-MIN .				
726	ILLEGAL COMMON DATA (AXIAL) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	LENGTH , WORK FACE and FIN-LENGTH do not satisfy the following condition: LENGTH ≥ WORK FACE + FIN-LENGTH > 0				
Action	Set correct data to LENGTH , WORK FACE and FIN-LENGTH .				
727	MATERIAL SHAPE CROSSING (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	There is a data overlap between the inside diameter side (IN) and outside diameter side (OUT) of the workpiece shape which has been designated in the MATERIAL unit.				
Action	Change the IN or OUT shape data.				
728	EXCESSIVE FINISH ALLOWANCE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The area to be rough-cut does not exist since the designated finishing allowance is larger than the shape defined by the sequence data.				
Action	Either change the finishing allowance or cancel rough-cutting.				
729	ILLEGAL SHAPE DESIGNATED (CNR) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the CORNER unit data, the relationship between the starting point and the ending point is wrong.				
Action	Set the correct data to SPT and FPT .				
730	ILLEGAL SHAPE DESIGNATED (EDG) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the FACING unit data, the relationship between the starting point and the ending point is wrong.				
Action	Set the correct data to SPT and FPT .				
731	ILLEGAL NUM. OF PATHS (THR) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The designated number of cutting times for #0, #3 thread type is less than 3.				
Action	Change the number of cutting times to 3 or more, or change the thread type.				
732	ACCELERATION DISTANCE EXCEED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The calculated distance of threading acceleration is in excess of the allowable value.				
Action	Change the related parameter setting value or reduce the surface speed to its minimum permissible value.				
733	ILLEGAL SHAPE DESIGNATED (GRV) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the T. GROOV unit data, the relationship between the starting point and the ending point is wrong.				
Action	Set the correct data to SPT and FPT .				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
734	ILLEGAL SHAPE EXCEEDS MTRL SIZE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	For machining of multiple grooves, the second and subsequent grooves are outside the workpiece profile defined by common unit.				
Action	Reduce the number of grooves.				
735	ILLEGAL DESIGNATED TL WID (GRV) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The designated tool width does not match to the groove shape specified in the T. GROOV unit.				
Action	Designate some other tool, or correct the width of the designated tool.				
736	ILLEGAL OVERLAP (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The parameter-set amount of grooving overlap is larger than the width or diameter of the designated tool.				
Action	Either designate some other tool, correct the width or diameter of the designated tool, or change the parameter setting (TC75).				
737	ILLEGAL DRILLING DIRECTION (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the T-DRILL or T-TAP unit data, the relationship between the starting point and the ending point is wrong.				
Action	Set the correct data to SPT-Z and FPT-Z .				
738	CORNER (R/C) DESIGNATED OVERLAP (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the unit (BAR, CPY, etc.), different types of corners (R and C) have been designated for portions that are identical in shape.				
Action	Check the designated corner portions and delete one of the corners.				
739	ILLEGAL FINISHING ALLOWANCE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the #0, #0 type of THREAD unit data, the following condition exists: (Finishing allowance) > (First depth-of-cut/4) Calculated from the height (HGT) and the number of times (NUMBER) in unit data. Specified by parameter TC78 .				
Action	Change either the thread height or the setting of parameter TC78 .				
740	ILLEGAL ANGLE IN FIRST SEQUENCE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The relationship between the thread shape and threading angle that are defined by the first sequence data of the THREAD unit is incorrect.				
Action	Change the threading angle, or change the coordinates of the starting or ending point of the first sequence.				
741	ILLEGAL THREAD ANGLE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In the machining type that requires the designation of a threading angle, the following condition is not satisfied. (Tool tip angle) ≤ (Threading angle)				
Action	Change either the threading angle or the data of the tool to be used.				

No.	Message (WNo., UNo., SNo.)	Type of error	Stopped status	Clearing procedure	Display
742	ILLEGAL THREAD HEIGHT In the THREAD unit data, the following condition is not satisfied. (Finishing allowance) ≤ (Thread height) Designated in the unit data. Set using parameter TC78 .	B	I (L)	O (S)	Red (Blue)
Action	Change the thread height or the setting of parameter TC78 .				
743	INTERSECTION NOT FOUND No automatic calculations of intersection were obtained since there are shape sequence data disparities.	B	I (L)	O (S)	Red (Blue)
Action	Correct the shape sequence data.				
744	DATA MISSING (INTERSECTION) No automatic calculations of intersection were obtained since there are lack of data to automatically calculate a point of intersection.	B	I (L)	O (S)	Red (Blue)
Action	Correct the unit data or the sequence data.				
745	INTERSECTION CALCULATE IMPOSS. No automatic calculations of intersection were obtained since there are illegal data in the program.	B	I (L)	O (S)	Red (Blue)
Action	Correct the unit data or the sequence data.				
746	NO DEPTH OF CUT INFO. A cutting depth cannot be determined for the tool since the DEPTH or CUT ANGLE data item on the TOOL DATA display is left blank.	B	I (L)	O (S)	Red (Blue)
Action	Set data in DEPTH or CUT ANGLE , or use some other tool.				
747	CHIP BREAKING CYCLE N/A Executing the chip-cutting cycle program has been attempted in spite of the chip-cutting cycle option being absent. (3 or 4 has been assigned to PAT . in the roughing tool sequence.)	B	I (L)	O (S)	Red (Blue)
Action	The chip-cutting cycle option must be present before the chip-cutting cycle program can be executed. (Assign either 0, 1, or 2 to PAT . in the roughing tool sequence.)				
748	CANNOT MAKE T-PATH (CHK DEPTH) 1. A normal path cannot be created since the cutting area is wider than twice the depth of cut per pass. 2. When the cutting area is equal to the clearance, a normal path cannot be created since the cutting area is wider than the depth of cut per pass. 3. A normal path cannot be created since the cutting area overlaps the deceleration distance and is wider than the depth of cut per pass.	B	I (L)	O (S)	Red (Blue)
Action	1. Increase the cutting depth to be set in the tool sequence, or narrow down the cutting area. 2. Increase the cutting depth to be set in the tool sequence, or reduce the clearance. 3. Two overlapping areas must not be present. Narrow down PRE-DIA (deceleration distance) to be set in the tool sequence, or narrow down the cutting area, or increase the cutting depth to be set in the tool sequence.				
749	—				
Cause	—				
Action	—				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
750	—				
Cause	—				
Action	—				
751	CURVE DEFINITION ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Blue
Cause	A curved surface that cannot be machined has been defined.				
Action	No corrective actions can be taken against this error; define a curved surface that can be machined.				
752	DESIGNATED AREA DATA IMPOSSIBLE (WNo., UNo., SNo.)	B	I (L)	O (S)	Blue
Cause	<p>The check surface values are incorrect.</p> <ol style="list-style-type: none"> For rough-machining 1 or 2: Check surface Z min. \geq material height For finishing: Check surface Z min. $>$ initial Z Check surface X min. $>$ X max., or Y min. $>$ Y max., or Z min. $>$ Z max. 				
Action	<p>Set the check surface values as follows:</p> <ol style="list-style-type: none"> For rough-machining 1 or 2: Check surface Z min. $<$ material height For finishing: Check surface Z min. \leq initial Z Check surface X min. \leq X max., and Y min. \leq Y max., and Z min. \leq Z max. 				
753	SMALL TOOL (WNo., UNo., SNo.)	B	I (L)	O (S)	Blue
Cause	In rough-machining 2, the tool diameter is extremely small in comparison with the dimensions of the defined 3-D figure.				
Action	Use tools whose diameters are no less than 1/100 of the distance between the maximum and minimum dimensions of the 3-D figure.				
754	LARGE TOOL (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Tool interference has occurred.				
Action	Set the approach path and the tool size so that the interference does not occur.				
755	R DIRECTION PITCH SMALL (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In rough-machining 2, the pitch in the radial direction is extremely small in comparison with the dimensions of the defined 3-D figure.				
Action	Set the radial-direction pitch to a value no less than 1/200 of the distance between the maximum and minimum dimensions of the 3-D figure.				
756	Z DIRECTION PITCH SMALL (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In rough-machining 2, the pitch in the Z direction is extremely small in comparison with the dimensions of the defined 3-D figure.				
Action	Set the Z-direction pitch to a value no less than 1/250 of (material height – height of the Z bottom of the 3-D figure).				
757	CURVE DEFINITION LARGE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> For rough-machining 2 with designation of workpiece size, the dimensions of the defined figure are larger than those of the workpiece. For rough-machining 2, a material height smaller than "(height of the Z bottom of the 3-D figure) + (height of a machining area outside the figure)" [parameters E84, E89] has been set irrespective of designating the offset amount or the workpiece size. 				
Action	Change the E84 and E89 parameter settings so that: for the offset amount designation, "(height of the bottom of the 3-D figure) + E84 < material height", and; for the workpiece size designation, "(height of the bottom of the 3-D figure) + E89 < material height".				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
758	INITIAL POINT SET ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In rough-machining 1 or 2, initial Z ≤ material height.				
Action	Change settings to give initial Z > material height.				
759	WPC UNIT NOT FOUND (MILL & TURN) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The workpiece shape cannot be defined since, in a program of workpiece scheme (MILL & TURN) with WPC setting, no WPC unit is prepared before machining units.				
Action	Create a WPC unit before the machining units and then enter the correct data.				
760	NO T. CENTER POINT IN WPC UNIT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	1. In a program of workpiece scheme (MILL & TURN) with WPC setting, the workpiece origin specified in the WPC unit which precedes C-axis point/line machining or turning units does not correspond to the axis of turning. 2. Additional basic coordinate system or work offset value is set in the WPC unit preceded by the common unit.				
Action	1. Before creating C-axis machining or turning unit, create a WPC unit with reference to the axis of turning. 2. Instead of additional basic coordinate system or work offset value, specify T. CENTER to set the WPC-zero point to the turning center in the WPC unit preceded by the common unit.				
761	HEAD ANGLE INCORRECT (C-AXIS) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In a program of workpiece scheme (MILL & TURN) with WPC setting, the machining section specified in a C-axis point/line machining or turning unit does not correspond to the B-axis angle specified in an index unit.				
Action	Before creating C-axis machining or turning unit, create an index unit to specify the B-axis angle appropriate for the section to be machined.				
762	PROGRAM IS NOT MILL & TURN TYPE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	It was attempted to set a turning or C-axis point/line machining unit in a program of initial-point scheme (ONLY MILL).				
Action	Delete the turning or C-axis point/line machining unit, or set it in a program of workpiece scheme (MILL & TURN) with WPC setting.				
763	START PROCESS ILLEGAL (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	It was attempted to execute restart operation from, or single-process operation of, an unsuitable unit.				
Action	Specify an appropriate unit for the desired mode of operation.				
764	MAIN PRG/SUB PRG TYPE MISMATCH (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	There are programming units of workpiece scheme with WPC setting and of the same scheme with Z-offset setting prepared in one and the same program.				
Action	Delete the incompatible units to create a program of the required scheme.				
765	Z-OFFSET NOT FOUND (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The indispensable Z-offset is not yet set for a program of workpiece scheme (MILL & TURN) with Z-offset setting.				
Action	Set the Z-offset item externally in the setup data page.				
766	MAXIMUM NO.OF LAYOUTS EXCEEDED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	It was attempted to run a program which contains a process with more than 4000 layout data items. The number of repetitions exceeds 9999990 for a program (CONTI. = 1 in the END unit) without workpiece transfer unit.				
Action	Reduce the number of layout data items for one process to no larger than 4000. Modify the program to reduce the number of repetitions to no larger than 9999990.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
767	ILLEGAL ESCAPE TOOL No. (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The setting in the parameter for the retraction tool is not correct.				
Action	Check the parameter concerned.				
768	4 AXIS MACHINING PROGRAM ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The setting for simultaneous machining is not correct. (Example : Only the upper or the lower turret is specified for simultaneous machining.)				
Action	Review the program and perform corrections as required (e. g. to change the turret to be used for the particular tool sequence of the unit concerned).				
769	ILLEGAL TOOL DIRECTION/SPDL ROT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The specification of the direction of turning spindle rotation differs between those tools on the upper and lower turrets which are to be used for simultaneous machining.				
Action	Select another tool, or change the specification in question, on either side for the same direction of turning spindle rotation as for the other tool.				
770	FIN LEN ERROR (CHECK COM UNIT) (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	For the execution of a composite program: The settings in the common unit do not satisfy the following condition: FIN-LENGTH ≤ LENGTH – WORK FACE.				
Action	Reduce the FIN-LENGTH setting to satisfy the above condition.				
771	ILLEGAL LOW TUR COMMON UNIT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A unit of machining with the lower turret (or including a command for its retraction) is created within a program whose common unit declares (under LOW TURR) that the lower turret is "not used".				
Action	Change over the LOW TURR setting, or modify the machining unit concerned in the turret selection.				
772	NO HEAD UNIT IN PROGRAM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	1. For the execution of a composite program: Machining units are not preceded by a head selection unit (HEAD). 2. For the execution of a composite program: The workpiece transfer unit (TRANSFER) is not followed by a HEAD unit.				
Action	1. Set a HEAD unit to specify the turning spindle to be used for the succeeding machining units. 2. Set a HEAD unit after the TRANSFER unit to specify the turning spindle to be used for the succeeding machining units.				
773	TRANSFER UNIT IN PROGRAM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	For the execution of an independent program: A workpiece transfer unit (TRANSFER) is erroneously set in a program to be run independently on either HD side.				
Action	Delete the inappropriate TRANSFER unit from the program.				
774	HEAD UNIT IN PROGRAM (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	For the execution of an independent program: A head selection unit (HEAD) is erroneously set in a program to be run independently on either HD side.				
Action	Delete the inappropriate HEAD unit from the program.				
775	TOOL CANNOT PERFORM FACING (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	For tool selection on a turret-type tool-post: The section to be machined by the selected tool differs from the corresponding setting in the unit data line.				
Action	Change the setting in the unit data line with respect to the tool, or select another tool as required.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
776	NO Y-AXIS (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A Y-axis machining unit is created for a turret not correspondingly executed.				
Action	Delete the inappropriate machining unit from the program, or describe the particular machining process without using the Y-axis control.				
777	INCORRECT POSIT. CANNOT RESTART (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The restarting function has been started from the balanced-cutting with the lower turret.				
Action	Specify the upper turret to execute the restarting function from balanced cutting.				
778	INCORRECT POSIT. CANNOT START (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The single-process function has been applied to the balanced-cutting with the lower turret.				
Action	Specify the upper turret to execute the single-process function for balanced cutting.				
779	— (, ,)				
Cause	—				
Action	—				
780	APPROACH PATH INTERFERENCE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The programmed shape of the approach path and/or retraction path interferes with the workpiece (programmed machining shape + machining allowance).				
Action	Reduce the amount of approach and/or the amount of overlap or the tool diameter. Or specify another position as the approach point.				
781	DBL SPDL OPER ILLEGAL TUR ASIGN (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In a 2 WORKPC unit the selection of the turret in the tool sequence does not correspond to the selection of the spindle (SP1 or SP2) in the unit data line.				
Action	Select the correct turret in the tool sequence data in question.				
782	DBL SPDL OPER NO TRANS UNIT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The 2 WORKPC machining unit is not preceded by a workpiece transfer unit.				
Action	Set a TRANSFER unit as required.				
783	DBL SPDL OP ILLEGAL SIMUL OP (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	There is an instruction of simultaneous machining given in the flow of "2 Workpiece" machining.				
Action	Clear the tool sequence concerned of the instruction of simultaneous machining.				
784	DBL SPDL OP ILLEGAL BALANCE CUT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	There is an instruction of balanced cutting given in the flow of "2 Workpiece" machining.				
Action	Clear the tool sequence concerned of the instruction of balanced cutting.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
785	DBL SPDL OPER ILLEGAL TUR ESC (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	There is an instruction of retraction given in the flow of "2 Workpiece" machining.				
Action	Clear the tool sequence concerned of the instruction of retraction.				
786	DBL SPDL OP ILLEGAL MEASUREMENT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	There is a measurement unit inserted in the flow of "2 Workpiece" machining.				
Action	Temporarily cancel the mode of "2 Workpiece" machining to execute the required in-process measurement.				
787	DBL SPDL OPER ILLEGAL TRANSFER (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	There is a workpiece transfer unit inserted in the flow of "2 Workpiece" machining.				
Action	Temporarily cancel the mode of "2 Workpiece" machining to execute the required workpiece transfer.				
788	DBL SPDL OPER ILLEGAL HEAD UNIT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The head selection unit in the mode of "2 Workpiece" machining has another setting than SINGLE under TYPE.				
Action	Temporarily cancel the mode of "2 Workpiece" machining to execute synchronous rotation of both spindles (by selecting SYNCH.).				
789	DBL SPDL OPER UNIT ERROR (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> For a flow of "2 Workpiece" machining an end instruction is given before the start instruction. For a flow of "2 Workpiece" machining a restart instruction is given before the end instruction. In a flow of "2 Workpiece" machining, only one spindle is specified in the head selection unit and machining instruction is given for only one spindle. 				
Action	<ol style="list-style-type: none"> Set the "2 Workpiece" machining units in correct order. Set the "2 Workpiece" machining units in correct order. Review the program for "2 Workpiece" machining. In a "2 Workpiece" machining unit, be sure to set two head selection units so that machining is performed with two spindles. 				
790	ILLEGAL BALANCE CUT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> Only a single line of tool sequence is created for balanced cutting. Tools different in shape (nose angle, infeed angle and nose R) are designated for the upper and lower turret in balanced cutting. 				
Action	<ol style="list-style-type: none"> Add another tool sequence line to use the other turret symmetrically for balanced cutting. Specify tools identical in shape (nose angle, infeed angle and nose R) for the upper and lower turret in balanced cutting. 				
791	MUST INPUT CUTTING SPEED (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The surface speed data set in the tool sequence contains an invalid value.				
Action	Set correct surface speed data in the tool sequence.				
792	MUST INPUT FEEDRATE (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The feed data set in the tool sequence contains an invalid value.				
Action	Set a correct feed data in the tool sequence.				

No.	Message (WNo., UNo., SNo.)	Type of error	Stopped status	Clearing procedure	Display
793	ILLEGAL MILL SPINDLE RPM If DRILL is selected as the turning drilling unit tool: 1. The milling spindle speed is less than the turning spindle speed when the rotation direction set in the tool sequence is the same as that of the milling spindle. 2. The milling spindle speed has exceeded the maximum rotation speed (SA1 to SA8) when the rotation direction set in the tool sequence is the same as that of the milling spindle. 3. The milling spindle speed is set to a negative value when the rotation direction set in the tool sequence is opposite to that of the milling spindle. 4. When the direction of rotation in the tool sequence is the opposite of the direction of mill spindle rotation, the turning spindle speed matches the relative speed calculated from the surface speed.	B	I (L)	O (S)	Red (Blue)
Cause	If DRILL is selected as the turning drilling unit tool: 1. The milling spindle speed is less than the turning spindle speed when the rotation direction set in the tool sequence is the same as that of the milling spindle. 2. The milling spindle speed has exceeded the maximum rotation speed (SA1 to SA8) when the rotation direction set in the tool sequence is the same as that of the milling spindle. 3. The milling spindle speed is set to a negative value when the rotation direction set in the tool sequence is opposite to that of the milling spindle. 4. When the direction of rotation in the tool sequence is the opposite of the direction of mill spindle rotation, the turning spindle speed matches the relative speed calculated from the surface speed.				
Action	1. Reduce the spindle speed for the turning spindle. 2. Set the milling spindle speed to a value lower than its maximum spindle speed. 3. Reduce the spindle speed for the turning spindle. 4. Reduce the spindle speed for the turning spindle.				
794	INCORRECT B AXIS POSITION For ATC (BA47 = 0) turret type machines with B-axis (BA48 = B), the direction of "part to be machined" set with the turning drilling unit does not match with the B-axis angle set with the index unit.	B	I (L)	O (S)	Red (Blue)
Cause	For ATC (BA47 = 0) turret type machines with B-axis (BA48 = B), the direction of "part to be machined" set with the turning drilling unit does not match with the B-axis angle set with the index unit.				
Action	Reset the B-axis angle for the index unit defined prior to setting the turning drilling unit to either 0 or 180°.				
795	SHIFT-Y TOO LARGE (, ,)				
Cause	When the amount of SHIFT-Y is not equal to zero in the mill-turning unit, the amount of SHIFT-Y is smaller than the radius of the tool or greater than (workpiece radius + tool radius).				
Action	Modify the program so that the amount of SHIFT-Y is equal to or greater than the radius of the tool or equal to or smaller than (workpiece radius + tool radius).				
796	NO MAZATROL L TURRET OPT (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt was made to execute a MAZATROL program with lower turret control enabled although the MAZATROL lower turret control option is unavailable.				
Action	The MAZATROL program cannot be executed because the MAZATROL lower turret control option is unavailable.				
797	BARRIER CANCEL ON (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The cycle start button has been pressed when the barrier was canceled.				
Action	Disable the "Cancel Barrier" setting.				
798	JAW NO NOT FOUND (WNo., UNo., SNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The cycle start button has been pressed when a jaw No. was not assigned.				
Action	Set the jaw No.				
799	— (, ,)				
Cause	—				
Action	—				

3-2-9 No. 800 - No. 899, No. 1800 - No. 1899 (EIA/ISO program error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display
800	— (, ,)				
Cause	<hr/>				
Action	<hr/>				
801	SIMULTANEOUS AXIS EXCEEDED (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The number of axis motion commands given in one block is in excess of the number of simultaneously controllable axes.				
Action	Modify the program with respect to the specification concerned.				
802	ILLEGAL AXIS NAME (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	1. The axis address names assigned in the program are different from those which have been parametrized. 2. Bit 4 of parameter M17 for the shaping control axis (the axis specified in parameter K3) is set to '0' (linear axis).				
Action	1. Correct the axis address names in the program. 2. Set bit 4 of parameter M17 for the shaping control axis (the axis specified in parameter K3) to '1' (rotational axis).				
803	DIVIDED COMMAND ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A distance of axis movement that cannot be divided by the preset command unit has been assigned.				
Action	Review the program.				
804	PARITY H ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	On paper tape, the number of holes per character is even for EIA code or odd for ISO code.				
Action	Check the paper tape and the tape reader.				
805	PARITY V ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	On paper tape, the number of holes per block is odd.				
Action	Make even the hole quantity per block on the paper tape; or turn off the bit 1 of the parameter TAP25 used for parity-V selection.				
806	ILLEGAL ADDRESS (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An address that is not covered in the specifications has been used.				
Action	Check and correct the corresponding address in the program, and also check the specifications.				
807	ILLEGAL FORMAT (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	1. The format in which the data has been designated in the program is incorrect. 2. Under the condition of workpiece placement error correction, or tool tip point control, an axis address other than those for the five controlled axes concerned (as specified in parameters K114 , K115 , K116 , K121 , and K125) was used in a motion command block.				
Action	1. Review the program. 2. Do not use an axis address other than those for the five controlled axes concerned (as specified in parameters K114 , K115 , K116 , K121 , and K125) in a motion command block under the condition of workpiece placement error correction, or tool tip point control.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
808	MIS-SET G CODE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A G-code that is not covered in the specifications has been designated.				
Action	Check and correct the corresponding G-code address in the program.				
809	ILLEGAL NUMBER INPUT (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The assigned data for the address is out of the allowable setting range.				
Action	Review the program.				
810	PROGRAM END NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	“EOR” has been detected during tape or memory operation.				
Action	For the main program, set M02 or M30 at the end of the program. For subprograms, set M99 at the end of the program.				
811	ILLEGAL O/N NUMBER (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Zeroes have been designated as program or sequence numbers.				
Action	Delete zero from N (sequence) or O (program) numbers of the program; or change O-No. (program numbers) to between 1 and 99999999, N-No. (sequence numbers) to between 1 and 99999.				
812	ERROR IN BUFFER BLOCK (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An error has been found to exist in the pre-read block during execution of tool diameter offset.				
Action	Review the program.				
813	INCH/METRIC OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The inch/metric selection command has been issued using the G-code although a G-code inch/metric selection function is not provided.				
Action	Check the specifications.				
814	INTERPOLATION OVERFLOW (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The specified moving distance is too long.				
Action	Specify a shorter distance.				
815	G60 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Program command G60 has been designated although a unidirectional positioning function is not provided.				
Action	Check the software specifications and change the program command G60 to G00.				
816	FEEDRATE ZERO (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The feedrate command has not been input.				
Action	Specify feedrate F for the movement command. (Since modal move command G01 is automatically set at power-on, axis movement in the modal mode is started by input of a move command, even if G01 is not designated in the program).				

3

ALARM

No.	Message (WNo., NNo., BNo.)	Type of error	Stopped status	Clearing procedure	Display
817	INCORRECT ARC DATA The relationship between the starting and ending points of the arc and the center of the arc is not appropriate.	B	I (L)	O (S)	Red (Blue)
<i>Cause</i>					
<i>Action</i>	Check the values of the starting/ending points and the address values of center of the arc in the program, and check the address values for the correct direction (minus or plus).				
818	MISSING CENTER (NO DATA) <ul style="list-style-type: none">• For arc interpolation by R designation, the coordinates of the center of the arc cannot be calculated.• Axis moving by manual interruption is specified in immediate front of the arc command. The arc cannot be defined since the relationship between the starting point, ending point, and radius of the arc becomes incorrect.	B	I (L)	O (S)	Red (Blue)
<i>Cause</i>					
<i>Action</i>	<ul style="list-style-type: none">• Correct the value of each address in the program.• Restart the program after pressing the RESET key.				
819	HELICAL OPTION NOT FOUND The helical interpolation command has been issued although such an interpolation function is not provided.	B	I (L)	O (S)	Red (Blue)
<i>Cause</i>					
<i>Action</i>	Check the specifications, and if such an interpolation function is not available, correct the data of the block in which the arc interpolation command has been issued with designation of three axes.				
820	G02.1, G03.1 OPTION NOT FOUND The spiral interpolation command (G02.1 or G03.1) has been issued although such an interpolation function is not provided.	B	I (L)	O (S)	Red (Blue)
<i>Cause</i>					
<i>Action</i>	Delete the G02.1 or G03.1 command.				
821	G07 OPTION NOT FOUND The virtual-axis command (G07) has been issued although there are not virtual-axis specifications.	B	I (L)	O (S)	Red (Blue)
<i>Cause</i>					
<i>Action</i>	Check the specifications, and then change the G07 command.				
822	ILLEGAL MODAL A G-code command has been specified in illegal modal data.	B	I (L)	O (S)	Red (Blue)
<i>Cause</i>					
<i>Action</i>	Review the program.				
823	G17 - G19 COMMAND IN M98 A plane selection command (G17, G18 or G19) has been issued during figure rotation.	B	I (L)	O (S)	Red (Blue)
<i>Cause</i>					
<i>Action</i>	Delete the plane selection command (G17, G18 or G19) from the figure rotation subprogram.				
824	G17 - G19 COMMAND IN G68 A plane selection command (G17, G18 or G19) has been specified in the coordinates rotation command (G68).	B	I (L)	O (S)	Red (Blue)
<i>Cause</i>					
<i>Action</i>	IF G68 has been issued, execute the coordinates rotation cancel command (G69) before specifying the plane selection command (G17, G18 or G19).				
825	G17 - G19 COMMAND IN G38 - G42 A plane selection command (G17, G18 or G19) has been specified during tool diameter offset (G41 or G42).	B	I (L)	O (S)	Red (Blue)
<i>Cause</i>					
<i>Action</i>	Specify the plane selection command after the tool diameter offset command has been canceled by G40.				

No.	Message (WNo., NNo., BNo.)	Type of error	Stopped status	Clearing procedure	Display
826	G95 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The synchronous feed command (G95) has been specified although such feed specifications are not provided.				
Action	After checking the specifications, change the synchronous feed command (G95) to the feed-in-minutes command (G94). Also change the F command value.				
827	F0 COMMAND IN G02, G03 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The F 1-digit rapid-feed command (F0) has been specified during arc interpolation (G02 or G03).				
Action	Since rapid feed cannot be ordered for arc interpolation, specify an F 1-digit command other than F0. Specify G0 or G1 if the type of interpolation is not arc interpolation.				
828	NO AUTO CORNER OVERRIDE OPTION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The automatic corner override command (G62) has been specified although such an override function is not available.				
Action	Check the specifications, and delete the G62 command from the program.				
829	ILLEGAL 2ND M CODE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The secondary auxiliary function address that has been specified in the program is different from the corresponding address that has been parameterized.				
Action	Check and correct the secondary auxiliary function address that has been specified in the program.				
830	G96 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The constant surface speed command (G96) has been specified although such specifications are not provided.				
Action	Check the specifications and change the constant surface speed command (G96) to the speed command (min^{-1}).				
831	G45,46,47,48 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A tool-position compensation command (G45 to G48) has been specified although such specifications are not provided.				
Action	Check the specifications.				
832	G45 - G49 COMMAND IN G98 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Tool-position compensation has been specified during figure rotation or coordinates rotation.				
Action	Review the program.				
833	1/4, 3/4 CIRCLES IN G45 - G48 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An arc command that is not available for tool-position compensation has been specified.				
Action	Review the program.				
834	G40, G41, G42 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A tool diameter offset command (G41 or G42) has been specified although such specifications are not provided.				
Action	Check the specifications.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
835	G41, G42, FORMAT ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A compensation command (G40, G41, G42) has been specified during the arc mode (G02 or G03 command).				
Action	Set either the linear command (G01) or the rapid-feed command (G00) into the compensation command block or the cancellation block. (That is, set the modal status to linear interpolation).				
836	NO INTERSECTION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In tool diameter offset (G41 or G42), the coordinates of the intersection point existing when a block was skipped in processing of interference blocks cannot be calculated.				
Action	Review the program.				
837	TOOL OFFSET INTERFERENCE ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An interference error has occurred during execution of tool diameter offset (G41 or G42).				
Action	Review the program.				
838	3-D OFFSET OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The three-dimensional compensation command has been designated although such compensation specifications are not provided.				
Action	Check the specifications.				
839	ILLEGAL OFFSET No. (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A compensation command (G41 or G42) has been designated without a compensation number (DOO); or the compensation number is larger than the maximum number of sets of compensation numbers available in the specifications.				
Action	Check the maximum available number of sets of compensation numbers, and designate a compensation number smaller than that.				
840	CANNED CYCLE OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A fixed-cycle G-code has been designated although fixed-cycle specifications are not provided.				
Action	Check the specifications and correct the program.				
841	DESIGNATED TOOL NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The tools that are not yet registered on the TOOL DATA display are specified using T-code command in an EIA/ISO program.				
Action	Check the program and register the necessary tools on the TOOL DATA display.				
842	SUB PROGRAM NESTING EXCEEDED (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> The total number of sequential calls of subprogram has exceeded eight. Executing a program that includes an "M99" command has been attempted in the direct operation mode of HD, IC memory card or the Ethernet. Invoking a program stored within the HD, IC memory card or the host has been attempted from the HD, IC memory card or the host. 				
Action	<ol style="list-style-type: none"> Check the number of subprogram calls, and correct the program so that the number of calls does not exceed eight. Do not include an "M99" command in the main program to be executed in the direct operation mode. Do not invoke any subprograms of the HD, IC memory card or host from the main program of the HD, IC memory card or the host. 				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
843	DESIGNATED Sno. NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The sequence number for subprogram call, for return from a subprogram or for the GOTO designation is not yet set.				
Action	Set the sequence number in the appropriate block.				
844	PROGRAM No. NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt was made to call a subprogram which was not yet registered.				
Action	Register the subprogram.				
845	ILLEGAL VARIABLE COMMAND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A variables number has been designated although variables number (#OO) specifications are not provided.				
Action	Check the specifications.				
846	DESIGNATED NUMBER NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The designated variables number is larger than the maximum variables number permitted by the specifications.				
Action	Check the specifications and the variables numbers in the program.				
847	NO "=" CODE IN PROGRAM (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	"=" was not designated in the definition of a variable.				
Action	Set "=" in the variables definition.				
848	M98 OPTION NOT FOUND (WNo., NNo., BNo.)	B	K	O	Blue
Cause	A figure rotation command has been designated although figure rotation specifications are not provided.				
Action	Check the specifications.				
849	FIGURE ROTATE NESTING EXCEEDED (WNo., NNo., BNo.)	B	K	O	Blue
Cause	One figure rotation command has been designated during execution of another such command.				
Action	Check the program.				
850	G68 AND M98 COMMANDS SAME BLOCK (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A figure rotation command and a coordinates rotation command are designated at the same time.				
Action	Check the program.				
851	G68 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The coordinates rotation command (G68) has been designated although coordinates rotation specifications are not provided.				
Action	Check the specifications.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
852	USER MACRO OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Macro specifications have been designated although such specifications are not provided.				
Action	Check the specifications.				
853	EXTERNAL MACRO OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A user macro interruption command has been designated although such interruption specifications are not provided.				
Action	Check the specifications.				
854	INCORRECT USERMACRO PROGRAMMING (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An NC statement and a macro statement are present in one block.				
Action	Review the program and give the NC statement and the macro statement in separate blocks.				
855	USER MACRO NESTING EXCEEDED (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The maximum permissible degree of multiplicity of user macro calls has been exceeded.				
Action	Review the program and correct it so that the number of user macro calls does not exceed the maximum number of calls permitted by the specifications.				
856	USER MACRO ARGUMENT EXCEEDED (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The number of sets of user macro call arguments of type II is too large.				
Action	Review the program.				
857	INCORRECT USER MACRO G67 PROG. (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Command G67 has been designated when G66 command modal state was not yet set.				
Action	The G67 command is the call cancellation command; after reviewing the program, designate firstly the G66 command and then the G67 command.				
858	USER MACRO "[" NESTING EXCEEDED (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The total number of "[" and "]" within one block has become more than five.				
Action	Review the program, and correct it so that the total number of "[" and "]" within one block does not exceed five.				
859	NUMBER OF PARENTHESIS MIS-MATCH (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The total number of "[" and "]" within one block differ.				
Action	Review the program, and correct it so that the total number of "[" and of "]" become the same.				
860	CALCULATION IMPOSSIBLE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The operation expression is not correct.				
Action	Review the program and correct the operation expression.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
861	DIVISION BY ZERO (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The denominator in the division expression is zero.				
Action	Review the program and correct it so that the denominator in the division expression does not become zero.				
862	INTEGER VALUE OVERFLOW (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The integral value has overstepped -2^{31} ($2^{31} - 1$) in the operation process.				
Action	Review the operation expression written in the program, and correct it so that after operation, the value of the integer does not overstep -2^{31} .				
863	REAL VALUE OVERFLOW (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The variables data is overflowing.				
Action	Review the variables data in the program.				
864	"IF" STATEMENT ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The statement of IF [<conditional expression>] GOTO is wrong.				
Action	Review the program.				
865	"WHILE" STATEMENT ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The statement of WHILE [<conditional expression>] DO ~ END is wrong.				
Action	Review the program.				
866	"SETVN" STATEMENT ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The SETVN statement containing the variables name is wrong.				
Action	Review the program and correct it so that the variables name in the SETVN statement consists of seven characters or less.				
867	DO-END NESTING EXCEEDED (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Of WHILE [<conditional expression>] DO ~ END, DO ~ END has appeared more than 27 times (degree of multiplicity).				
Action	Review the program and correct it to reduce the degree of multiplicity of DO ~ END to no larger than 27 (27 times).				
868	DO-END MIS-MATCH (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The DO's and END's are not formed in pairs.				
Action	Review the program and correct it to give DO's and END's in pairs.				
869	NO USER MACRO IN TAPE MODE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	During tape operation, macro command (WHILE, DO, END, IF, GOTO, POPEN, PCLOS, BPRNT or DPRNT) has been found to exist in the tape contents.				
Action	Execute the program in the memory operation mode since blocks containing a macro command (WHILE, DO, END, IF, GOTO, POPEN, PCLOS, BPRNT or DPRNT) cannot be executed during tape operation.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
870	ILLEGAL VARIABLE NAME (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The designated variables name is not correct.				
Action	Review the variables names in the program and correct the corresponding variables name.				
871	VARIABLE NAME EXISTS (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	There are overlapping variables names.				
Action	Correct the program so that variables names do not overlap.				
872	G51 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A scaling command (G50 or G51) has been designated although scaling specifications are not provided.				
Action	Check the specifications.				
873	G51.1 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A mirror image command (G50.1 or G51.1) has been designated although programmable mirror image specifications are not provided.				
Action	Check the specifications.				
874	CORNER R/C OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Corner chamfering or corner rounding I/II has been designated although such specifications are not provided.				
Action	Check the specifications and delete corner rounding or corner chamfering from the program.				
875	NOT FOUND GEOMETRIC OPTION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The geometric command has been designated although geometric specifications are not provided.				
Action	Check the specifications.				
876	NOT FOUND GEOMETRIC OPTION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The geometric setting format is wrong.				
Action	Review the program.				
877	G15 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The polar coordinates command (G16) has been designated although such command specifications are not provided.				
Action	Check the specifications.				
878	ADDRESS CHANGE OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Absolute/incremental axis address conversion has been designated although such conversion specifications are not provided.				
Action	Check the specifications.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
879	G10 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Program data input has been designated although such input specifications are not provided.				
Action	Check the specifications.				
880	AXIS NOT ZERO RETURNED (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A move command other than that for reference-point return has been designated for the axis that was not returned to its reference point.				
Action	Manually return the axis to its reference point.				
881	G30 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Second, third or fourth reference-point return has been designated although such reference-point returning specifications are not provided.				
Action	Check the specifications.				
882	ILLEGAL COMMAND CROSS MACHINING (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The program section under the mode of cross machining contains an incompatible G-code.				
Action	Review the program.				
883	ILLEGAL CROSS MACHINING COMMAND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	1. A G110 command is given under modal conditions not suitable to cross machining control. 2. An unsuitable axis (since it cannot be used successfully for the counterpart) is specified for cross machining control.				
Action	Review the program.				
884	REFERENCE POINT RETURN CHECK (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An axis had not returned to the zero-point when the zero-point check command (G27) was executed.				
Action	Review the program.				
885	G22 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The before-movement stroke check function (G22) has been designated although such function specifications are not provided.				
Action	Check the specifications.				
886	BEYOND AREA OF G22 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	This alarm message is displayed before execution of a movement block to indicate that the ending point of the axis movement designated in the block is likely to enter the forbidden area which has been designated using the before-movement stroke check function (G22).				
Action	Review the axis-address coordinate values in the program.				
887	TAPE I/O ERROR (WNo., NNo., BNo.)	B (G)	I	O	Red (Blue)
Cause	1. Errors have occurred in the tape reader or printer errors have occurred during macroprogram data printing. 2. Host computer program used for Ethernet operation has failed.				
Action	1. Check for parameter errors. 2. Check for improper connection between the host computer containing the designated program, and the NC unit.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
888	FILE I/O ERROR (, ,)	E	I	O	Red (Blue)
Cause	The machining program file cannot be read.				
Action	Contact Mazak Technical Center or Technology Center.				
889	G37 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The automatic tool-length measurement command (G37) has been designated although such measurement specifications are not provided.				
Action	Check the specifications.				
890	G31 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The skip command (G31) has been designated although skip specifications are not provided.				
Action	Check the specifications.				
891	G31.1 - G31.3 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A multi-step skip command (G31.1, G31.2 or G31.3) has been designated although such skip specifications are not provided.				
Action	Check the specifications.				
892	AUTO PROGRAMMING FAILURE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A trouble has occurred with the software of auto program during the operation.				
Action	Contact Mazak Technical Center or Technology Center.				
893	PROGRAM DATA MISSING (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	Argument P was not designated in the macro call command (G65, G66, G66.1).				
Action	Review the program and set the number of the macro program to be called to argument P.				
894	MAZATROL PROGRAM DESIGNATED (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	1. An attempt has been made to call a MAZATROL program from an EIA/ISO program which was designated as a subprogram of MAZATROL program. 2. A MAZATROL program has been specified using G65 command (subprogram call) in the manual program mode unit of the MAZATROL program.				
Action	In cases 1 and 2 above, a MAZATROL program cannot be called as a subprogram. Review the program.				
895	IC CARD I/O BUSY (, ,)	A	I	O	Red
Cause	An attempt has been made to execute the IC memory card operation during data I/O operation with an IC memory card.				
Action	Execute the IC memory card operation after stop or completion of the data I/O operation with an IC memory card.				
896	ILLEGAL CHAMFERING (, ,)	B	J	P	Red
Cause	1. In the thread cutting cycle command, designation for chamfering is incorrect. 2. The current plane differs from the plane of chamfering.				
Action	1. Set chamfering data that ensures termination of the operation of the tool within the cycle. 2. Select the same plane as that of chamfering.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
897	LAP CYCLE BLOCK NUMBER EXCEED (, ,)	B	J	P	Red
Cause	The number of blocks in the shape data of the turning fixed-cycle (G270 through G273) exceeds 100 blocks.				
Action	Reduce below 100 the number of blocks in the shape data of the turning fixed-cycle (G270 through G273).				
898	LAP CYCLE ILLEGAL SHAPE DESIGN. (, ,)	B	J	P	Red
Cause	The shape defined in the turning fixed-cycle (G270 through G273) is not the shape for correct cutting.				
Action	Recheck the shape data specified by the turning fixed-cycle (G270 through G273).				
899	ILLEGAL TAPER LENGTH (, ,)	B	J	P	Red
Cause	In the fixed-cycle (G290, G294), designation for the taper length or the taper height is incorrect.				
Action	Set the taper length or the taper height for the fixed-cycle (G290, G294) smaller than the travel of the axis.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1800	CANNOT CHANGE TIME CONSTANT (WNo., NNo., BNo.)	B	I	P	Red
Cause	The G-command has been set in the block containing a time constant changeover/cancellation command. The time constant changeover/cancellation command has been set in the fixed cycle. The time constant changeover/cancellation command has been set during MAZATROL programmed simultaneous machining or MAZATROL programmed "2 workpiece" machining.				
Action	Review and correct the program.				
1801	ILLEGAL COMMD TIME CONST. CHANGE (WNo., NNo., BNo.)	B	I	P	Red
Cause	The G10 command has been set during time constant changeover (non-M880 mode).				
Action	Review and correct the program.				
1802	LLEGAL STARTUP CONDITION G12.1 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	1. The G12.1 command is set when the current position of the No. 1 axis (linear axis) on the plane is present on the negative side. 2. The No. 2 axis (rotational axis) on the plane is not a rotating type of rotational axis.				
Action	1. Move the current position of the No. 1 axis (linear axis) on the plane into the positive side before setting the G12.1 command. 2. Specify a rotating type of rotational axis as the No. 2 axis on the plane. Example : G17X_C_ ; G12.1 ; ...				
1803	ILL GCODE (2ND SPDL POS ONLY) (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A G-code whose usage for the secondary spindle is prohibited has been entered when the secondary spindle high-accuracy index option was valid.				
Action	When the secondary spindle high-accuracy index option is valid, an interpolation command cannot be issued for the secondary spindle; only a positioning command can be used.				
1804	NO Z-OFFSET (WNo., NNo., BNo.)	B	I	P	Red
Cause	With F165 bit 6 = 1 (Z-offset data presence check for EIA automatic operation), Z-offset value was missing in setup data when the G53.5 command (initial G53.5 included) was entered for EIA automatic operation.				
Action	Set the Z-offset value in the setup data or change the setting of F165 bit 6 from 1 to 0.				
1805	G68.2 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Executing the program has been attempted despite an inclined-surface machining option being absent.				
Action	The program for inclined-surface machining cannot be executed without the inclined-surface machining option.				
1806	ILLEGAL CMD TILTED PLANE CMD (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A command (G-command or T-command) that cannot be used for inclined-surface machining has been specified during inclined-surface machining.				
Action	Cancel inclined-surface machining before executing a G-command or T-command that cannot be used during inclined-surface machining.				
1807	CANNOT USE G68.2 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	1. Inclined-surface machining has been specified in the mode that prohibits the specification of inclined-surface machining. 2. A G68.4 command (for incremental coordinate system establishment) was given with the inclined-surface machining mode being cancelled.				
Action	1. Cancel the mode that prohibits the specification of inclined-surface machining. 2. With the inclined-surface machining mode being cancelled, use G68.2, instead of G68.4, to establish a particular coordinate system for ease of programming.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1808	CANNOT USE G53.1 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Inclined-surface machining has been specified in the mode that prohibits the specification of tool axial control.				
Action	Cancel the mode that prohibits the specification of tool axial control.				
1809	TILTED PLANE CMD FORMAT ERROR (WNo., NNo., BNo.)	A	L	S	Blue
Cause	The inclined-surface machining command address is wrong.				
Action	Correct the program.				
1810	TILTED PLANE CANNOT BE DEFINED (WNo., NNo., BNo.)	A	L	S	Blue
Cause	The inclined surface cannot be defined with the specified data.				
Action	Correct the program.				
1811	ILLEGAL CMD IN G43.1 (WNo., NNo., BNo.)	B	L	O	Red
Cause	"Tool length correction in the tool axial direction" has been specified in the mode that does not permit that command.				
Action	Cancel modes that cannot be used with "Tool length correction in the tool axial direction".				
1812	CANNOT USE G43.1 (WNo., NNo., BNo.)	B	L	O	Red
Cause	An incorrect command (such as a G-command or a T-command) has been entered during "Tool length correction in the tool axial direction".				
Action	Cancel "Tool length correction in the tool axial direction" to execute the G- or T-command.				
1813	G54.4 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The workpiece placement error correction option is not present.				
Action	The workpiece placement error correction cannot be used without the workpiece placement error correction option.				
1814	ILLEGAL CMD IN G54.4 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An incorrect command (such as a G-command or a T-command) has been entered during the workpiece placement error correction.				
Action	Cancel the workpiece placement error correction to execute the G- or T-command. With parameter F156 bit 2 set to 1, even when the movement is separated between rotary axis and linear axis during the workpiece placement error correction, this separation is not possible if tool radius compensation is used concurrently. With parameter F156 bit 2 set to 1, when the movement of rotary axis and linear axis is separated during the workpiece placement error correction, the MSTB command cannot be included in the same block.				
1815	CANNOT USE G54.4 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The workpiece placement error correction has been specified in the mode that does not permit the workpiece placement error correction.				
Action	Cancel modes that cannot be used with the workpiece placement error correction.				
1816	CANNOT USE G18.1/G18.2/G18.3 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The mode existing during plane selection (with G18.2/G18.3) for turning, or during plane cancellation (with G18.1) for turning is not valid.				
Action	Review and correct the program.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1817	ILLEGAL CMD IN G18.2/G18.3 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An incorrect command has been specified during the plane selection (with G18.2/G18.3) for turning.				
Action	Review and correct the program.				
1818	ILLEGAL COMMAND (SAFETY SHIELD) (, ,)	B	L	N	Red
Cause	An invalid cutting command was issued for the material model when interference checking of the material model was valid.				
Action	Specify the material model cutting invalid M-code in the program.				
1819	ILLEGAL C/S (SAFETY SHIELD) (, ,)	B	L	N	Red
Cause	<ol style="list-style-type: none"> The program of either the hard disk operation or Ethernet operation area was specified as subprogram when automatic operation interference check was valid. The program to be called up as a subprogram is not registered in the memory. 				
Action	<ol style="list-style-type: none"> Make the Automatic Operation INTELLIGENT SAFETY SHIELD function invalid, or before specifying the program of either the hard disk operation or Ethernet operation area as subprogram, set the INTELLIGENT SAFETY SHIELD function invalid M-code in the main program, to make the automatic operation interference check function invalid. Register the required program. 				
1820	ILLEGAL COMM IN G54.4 (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> Such a command has been assigned that the travel on a rotational axis exceeds an angle of 180° before or after the correction of the tool direction. The distance of correcting motion on the primary or secondary rotational axis for workpiece placement error correction is beyond the limit particularly set for MAZATROL programs. 				
Action	<ol style="list-style-type: none"> Divide the move command so that the angular motion, whether it be before or after being corrected, will be up to 180° in terms of the travel per block. Modify the settings of angular errors (Δa, Δb, Δc) in order that the distance of correcting motion on the primary or secondary rotational axis may not exceed the limit concerned. 				
1821	UNWRITABLE SYSTEM VARIABLE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Writing into a write-protected system variable has been attempted.				
Action	Correct the program.				
1822	— (, ,)				
Cause	—				
Action	—				
1823	NOT POSSIBLE SETTING ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The workpiece setup error correction command has been issued in a MAZATROL program under inadequate conditions.				
Action	<p>Confirm the followings:</p> <ul style="list-style-type: none"> Parameter K113 = 1 (Machine type: Tool tilt type) Workpiece setup error correction for MAZATROL program (option) valid 				
1824	— (, ,)				
Cause	—				
Action	—				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1825	— (, ,)				
Cause	—				
Action	—				
1826	HEAD-AXIS ILLEGAL COMMAND (WNo., NNo., BNo.)	B	I	O (S)	Red (Blue)
Cause	A G-code command that cannot be used for the head indexing axis has been specified.				
Action	Review and correct the program. Use G00 to index the head.				
1827	HEAD-AXIS ANGLE INCORRECT (WNo., NNo., BNo.)	B	I	O (S)	Red (Blue)
Cause	The angular value given for indexing the head cannot be divided by the L17 parameter setting (minimum angle of head index command).				
Action	Review and correct the program.				
1828	— (, ,)				
Cause	—				
Action	—				
1829	NO 5 AXIS SPLINE CUTTING OPTION (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	Tool tip point control or high-speed smoothing control is made valid in the G61.2 mode when the 5-axis spline interpolation option is not provided.				
Action	When the 5-axis spline interpolation is not provided, 5-axis spline interpolation is not available.				
1830	CANNOT USE 5AXIS SPLINE CUTTING (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The 5-axis spline interpolation mode is made valid during the uniaxial rotation interpolation (when bit 3 of parameter F85 is set to 0).				
Action	To use the 5-axis spline interpolation, select the joint interpolation (set bit 3 of parameter F85 to 1).				
1831	ILLEGAL CHOPPING CONDITION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> 1. The top dead point and the bottom dead point are the same. 2. The chopping feed rate command specifies zero. 3. The chopping command is given in the chopping command inhibiting mode. 4. An illegal value (other than 0 to 2) is set for the chopping offset method. 5. An illegal value (other than 0 to 7) is set for the chopping offset number. 6. The chopping record M-code is issued when the offset update method is selected. 7. An overflow occurs when writing the chopping speed value. 				
Action	<p>Take action below to remove the cause, and press the reset key or clear key.</p> <ol style="list-style-type: none"> 1. Review the command for the top dead point or bottom dead point. 2. Review the chopping feed rate command. 3. Issue the chopping command in the chopping command possible mode. 4. Set a correct value (0 to 2) for the chopping offset method. 5. Set a correct value (0 to 7) for the chopping offset number. 6. Do not issue the chopping record M-code when the offset update method is selected. 7. Specify the chopping speed value so that an overflow will not occur during writing. 				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1832	ILLEGAL COMMAND IN CHOPPING (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A command that is inhibited during chopping is issued.				
Action	Change the program so that the inhibited command is not issued during chopping. Then press the reset key or clear key.				
1833	ILLEGAL COND. CORRECT LENGTH.C (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<p>Tool length compensation G43P3 is issued under any of the following conditions.</p> <ol style="list-style-type: none"> Pinch cut or Y-axis turning option is invalid. The machine is not equipped with tool length C compensation, or tool length C compensation is made invalid. (Parameter BA131 bit 1 = 0) MAZATROL tool length is invalid. (Parameter F93 bit 3 = 0) Tool length A, B or C is not set for the current tool in MAZATROL tool data. 				
Action	<ol style="list-style-type: none"> Pinch cut or Y-axis turning option must be purchased. If the machine is not equipped with tool length C compensation, correct the program so that the G43P3 command is not used. If the machine is equipped with tool length C compensation, make tool length C compensation valid. (Parameter BA131 bit 1 = 1) Make MAZATROL tool length valid. (Parameter F93 bit 3 = 1) Set tool length A, B and C for the current tool in MAZATROL tool data. 				
1834	CORRECTING LENGTH.C (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> The X-, Y- or Z-axis movement command is issued before the head rotary axis (B-axis) is turned to the 90-degree position after the tool length compensation G43P3 command was issued (or after tool change to a tool with tool length C setting). A G-code command that is prohibited in the tool length compensation G43P3 mode is issued. 				
Action	<ol style="list-style-type: none"> When the axis movement command is issued with tool length C compensation valid, the head rotary axis (B-axis) must be turned to the 90-degree position in advance. Review the program and delete prohibited G-codes. 				
1835	FILE ACCESS ERROR (, ,)	G	I	P	Red
Cause	Access to the program file used for the hard disk or Ethernet operation did not complete within the preset time.				
Action	Contact Mazak Technical Center or Technology Center.				
	(, ,)				
Cause					
Action					

3-2-10 No. 900 - No. 999, No. 1900 - No. 1999 (EIA/ISO program error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display
900	— (, ,)				
Cause	—				
Action	—				
901	INCORRECT FIXED CYCLE COMMAND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The fixed-cycle command has been set in the program during the tool diameter offset (G41 or G42) modal status.				
Action	Set the tool diameter offset cancellation command (G40) before the fixed-cycle command.				
902	G10 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The G10 command has been designated although this command is not available with the system.				
Action	Check the specifications.				
903	ILLEGAL G10 L NUMBER (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An unallowable L number has been designated during input of G10 program command.				
Action	Correct the L number in the program.				
904	ILLEGAL G10 OFFSET No. (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Compensation numbers other than the number of sets permitted by the specifications have been designated during input of G10.				
Action	After checking the number of compensation sets permitted by the specifications, change the setting of address P to a value smaller than the permissible number of sets.				
905	G11 OPTION NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The G11 command has been designated although this command is not available with the system.				
Action	Check the specifications.				
906	NO S DIRECTIVE IN FIXED CYCLE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The spindle speed for the fixed cycle has not yet been set in the program.				
Action	Program the spindle speed command in the block which precedes the block with the fixed cycle command.				
907	DIFFERENT SPINDLE TYPE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to machine the workpiece using the synchronous tapping method in spite of the spindle controller being an SE type.				
Action	Use the appropriate tapping method for the particular type of the spindle controller.				
908	NO PITCH IN FIXED CYCLE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The pitch or the number of threads has not been designated for the tapping cycle (G74 or G84) of the drilling fixed cycles.				
Action	Designate the pitch using address F or E.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
909	ILLEGAL PITCH IN FIXED CYCLE (WNo., NNo., BNo.)	B	K	S	Blue
Cause	The pitch or the number of threads designated for the tapping cycle (G74 or G84) of the drilling fixed cycles is wrong.				
Action	Check and correct the pitch or the number of threads.				
910	— (, ,)				
Cause	—				
Action	—				
911	CORNER R/C OPTION NOT FOUND (WNo., NNo., BNo.)	B	K	S	Blue
Cause	Corner chamfering/corner rounding has been designated although such specifications are not provided.				
Action	Check the specifications and delete corner rounding or corner chamfering from the program.				
912	NO MOTION COMMAND AFTER R/C (WNo., NNo., BNo.)	B	K	S	Blue
Cause	The block that is to succeed the corner rounding or corner chamfering command does not consist of a move command.				
Action	Give the G01 command in the corresponding block.				
913	INCORRECT R/C COMMAND (WNo., NNo., BNo.)	B	K	S	Blue
Cause	The length of the corner rounding or corner chamfering that has been designated in the corner rounding or chamfering command is larger than the distance of movement.				
Action	Reduce the length of the corner rounding or chamfering to a value smaller than the distance of movement.				
914	INCORRECT COMMAND AFTER R/C (WNo., NNo., BNo.)	B	K	S	Blue
Cause	The movement distance designated in the next block is shorter than the length of the corner rounding or corner chamfering.				
Action	Reduce the length of the corner rounding or chamfering to a value smaller than the moving distance of the next block.				
915	ANGLE < 1 DEGREE (WNo., NNo., BNo.)	B	K	O	Blue
Cause	In the geometric command, the difference in angle between the two straight lines which intersect with each other is less than 1°.				
Action	Increase the angle difference in the geometric command.				
916	GEOMETRIC COMMAND NOT ABSOLUTE (WNo., NNo., BNo.)	B	K	O	Blue
Cause	The second block of the geometric command is an incremental command.				
Action	The second block must always consists of absolute data. Program it in units of absolute coordinates.				
917	NO LINEAR COMMAND IN 2ND BLOCK (WNo., NNo., BNo.)	B	K	O	Blue
Cause	The second block of the geometric command is not given the linear command (G1).				
Action	Correct the program so that the linear command (G1) and the feedrate command (F) are given to the second block.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
918	INCORRECT ANGLE DATA (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	In address designation of the geometric command, the angle in the first block, ending point coordinates and angle in the second block are incorrectly given.				
Action	Check and reprogram the corresponding data.				
919	INCORRECT PLANE SELECTION CMD. (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A plane selection command (G17, G18 or G19) was given in the geometric command block.				
Action	Program the plane selection command (G17, G18 or G19) in the block that precedes the geometric command block.				
920	G27, M COMMANDS SAME BLOCK (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An M independent command (M0, M1, M2 or M30) has been programmed in the same block as the G27 command.				
Action	Correct the program so that the G27 command and the M independent command are contained in separate blocks.				
921	G29, M COMMANDS SAME BLOCK (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An M independent command (M0, M1, M2 or M30) and the G29 command (start-position return) have been programmed in the same block.				
Action	Correct the program so that the G29 command and the M independent command are contained in separate blocks.				
922	SKIP SPEED ZERO (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The feedrate F has not been programmed in the G31 (skip) command block.				
Action	Set the skip feedrate F into the G31 program block.				
923	ILLEGAL COMMAND G37 AXIS (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	No axis settings are included in the automatic tool-length measurement block; or more than one axis setting have been made.				
Action	Designate only one axis.				
924	G37, H COMMANDS SAME BLOCK (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The H-code is in the same block as the automatic tool-length measurement command.				
Action	Set the H-code into a block preceding the automatic tool-length measurement block.				
925	H CODE REQUIRED (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The H-code is not yet set for automatic tool-length measurement.				
Action	Set an H-code into a block preceding the automatic tool-length measurement block.				
926	ILLEGAL G37 SIGNAL (WNo., NNo., BNo.)	B	I	O	Red
Cause	The signal of measuring-position arrival has been turned on before the tool reaches the area designated through either a D-code or the parameter for deceleration area "d"; or the signal has not been turned on at all.				
Action	Check the program and parameters.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
927	SKIP COMMAND IN CORRECTING DIA (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The skip command (G31) was given during tool diameter offset (G41 or G42).				
Action	Correct the program so that the skip command is executed only after the cutter-diameter compensation cancellation command (G40) has been executed.				
928	ILLEGAL HEAD DATA No. (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The offset number that has been designated in the program is outside the range from 0 to 10.				
Action	Review the machining program and set an allowable offset number.				
929	HEAD DATA No. NOT FOUND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An "H_" number is missing in the "G45.1H_" part of the EIA/ISO program.				
Action	Review the machining program and set an allowable offset number.				
930	ILLEGAL HEAD TYPE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The head type does not match to the face that has been designated in the program.				
Action	Review the machining program and set the correct head type.				
931	NO HEAD DATA (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The head number that has been designated in the program is not registered on the HEAD OFFSET display.				
Action	1. Review the designated head number. 2. Check if the designated head number is registered on the HEAD OFFSET display.				
932	RETURN R POINT IN CUTTING SIDE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Return to reference point has been designated during the surface-machining mode (G17.2 to G17.5) of the program.				
Action	Return to reference point cannot be executed during the surface-machining mode (G17.2 to G17.5). Review the cutting program.				
933	NO 5FACE CUTTING OPTION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to execute a five-surface machining program when the five-surface machining option was not present.				
Action	Set the five-surface machining option to execute a five-surface machining program.				
934	NO HIGH-SPEED MODE OPTION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to execute the high-speed mode program when the high-speed mode option was not set.				
Action	Without the high-speed mode option, the high-speed mode program cannot be executed.				
935	NO PRE-INTERP ACCEL/DECEL OPT. (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to execute the high-accuracy mode program when the high-accuracy mode option was not set.				
Action	Without the high-accuracy mode option, the high-accuracy mode program cannot be executed.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
936	OPTION NOT FOUND (WNo., 0, 0)	E	I (L)	O	Red
Cause	Either of the following seven options is missing (identify the corresponding option from the work number displayed in parentheses): 1. NURBS interpolation option 2. Shaping option 3. Planet tapping option 4. Shape correction control option 5. Auto pecking cycle of the cutting load detection type 6. Ethernet operation 7. Cylinder interpolation option				
Action	Check the specifications.				
937	ILLEGAL TOOL DESIGNATED (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The designated tool cannot be used.				
Action	Check the TOOL DATA display to see if the designated tool is an unusable one.				
938	NO IC CARD MODE OPTION (, ,)	A	K	P	Red
Cause	An attempt has been made to execute the IC memory card operation although the optional function of IC memory card operation is not available.				
Action	This operation cannot be executed because the optional function of IC memory card operation is not available.				
939	NO THREAD CUTTING OPTION (WNo., NNo., BNo.)	A (A)	K (L)	P (S)	Red (Blue)
Cause	An attempt has been made to execute operation or tool path check of the program that contains G33 command (threading), although G33 option is not provided.				
Action	Without G33 option, G33 threading command cannot be used.				
940	NO INVERSE TIME OPTION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	Inverse time feed program was attempted although inverse time feed option is not provided.				
Action	Inverse time feed program cannot be executed because inverse time feed option is not provided.				
941	G93 MODE (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	G-code of inhibition during G93 mode has been designated.				
Action	Review the program and delete prohibited G-codes.				
942	NO 3-D CONVERSION OPTION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to execute the three-dimensional coordinate conversion program in the absence of a three-dimensional coordinate conversion option.				
Action	Three-dimensional coordinate conversion is not possible because of the absence of a three-dimensional coordinate conversion option.				
943	CONVERTING IN 3-D COORDINATES (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An illegal G-code in the G68 mode has been designated.				
Action	Review the program, and delete the illegal G-code.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
944	WRONG CMD. IN 3-D COORDINATES (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A G68 command has been designated during the modal information that does not permit G68 to be set.				
Action	Review the program, and modify the modal information existing when the G68 command was designated.				
945	NO HV MACHINING FUNC. OPTION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to execute A-axis/B-axis automatic, sequential control or A-axis direct programming in the absence of an HV machining option.				
Action	Neither A-axis/B-axis automatic, sequential control, nor A-axis direct programming is possible because of the absence of the option.				
946	NO MAZ. SUB PROGRAM OPTION (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to call up a MAZATROL program from the EIA/ISO program in spite of the absence of a MAZATROL call option.				
Action	Since a MAZATROL call option is not present, MAZATROL programs cannot be called up from EIA/ISO programs using the subprogram call function.				
947	NO BLOCK SKIP OPTION (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to carry out block skip operations in spite of the absence of a block skip option.				
Action	Block skipping is not possible because of the absence of a block skip option.				
948	NO G54.1 OPTION (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to use a G54.1 code in spite of the absence of a G54.1 (additional workpiece coordinate system) option.				
Action	A G54.1 code cannot be used because of the absence of a G54.1 (additional workpiece coordinate system) option.				
949	NO G52 IN G54.1 MODE (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to use an additional workpiece coordinate system and a local workpiece coordinate system at the same time.				
Action	An additional workpiece coordinate system and a local workpiece coordinate system cannot be used at the same time.				
950	NO SPLINE CUTTING OPTION (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to specify a spline interpolation command in spite of the absence of a spline interpolation option.				
Action	A spline interpolation command cannot be used because of the absence of a spline interpolation option.				
951	NO CORNER C/R COMMAND IN G0/G33 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	A corner chamfering/rounding command has been designated in the G0 or G33 mode.				
Action	A corner chamfering/rounding command cannot be designated in the G0 or G33 mode.				
952	NO SYNCHRONIZED TAP OPTION (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An attempt has been made to perform synchronized tapping in spite of the absence of a synchronized tapping option.				
Action	Synchronized tapping cannot be performed because of the absence of a synchronized tapping option.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
953	TOOL DATA INPUT PROCESS ERROR (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	During EIA/ISO program or MDI execution, data is not entered in the tool data item LENGTH (TOOL SET X, TOOL SET Z) or ACT-φ (NOM-φ) . (Only when the MAZATROL tool length and tool diameter are valid and parameter F84 bit 7 = 1)				
Action	Review the tool data and enter necessary data if missing. (Related parameters: F84 bit 7, F92 bit 7, F93 bit 3) Enter a value in ACT-φ (NOM-φ) if the MAZATROL tool diameter is valid (F92 bit 7 = 1). Enter a value in LENGTH (TOOL SET X, TOOL SET Z) if the MAZATROL tool length is valid (F93 bit 3 = 1). If F84 bit 7 = 0, any missing data is ignored and the program and MDI are executed.				
954	SCREW PITCH ERR (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	The thread lead (thread pitch) that has been designated in the threading command data is not correct.				
Action	Set the correct thread lead in the threading command data.				
955	START AND END POINT NOT AGREE (WNo., NNo., BNo.)	E	I (L)	O	Red
Cause	The ending point of the block immediately preceding the G06.2 command data, and the command data in the starting block of G06.2 do not match.				
Action	Modify the program so that the coordinate command data in the starting block of G06.2 matches the ending point of the immediately preceding block.				
956	RESTART OPERATION NOT ALLOWED (WNo., NNo., BNo.)	E	I (L)	O	Red
Cause	<ol style="list-style-type: none"> Restarting from the block containing the G06.2/G43.4 mode data has been attempted. During the control of cross machining, restarting the program containing another cross machining command has been attempted in the corresponding command sequence or subsequent. During the INTELLIGENT SAFETY SHIELD for automatic operation is valid, EIA modal restarting from the block preceded by the M660 block has been attempted. When 1 is set for bit 3 of parameter F93 or bit 7 of parameter F94 with a turning tool mounted, restarting has been attempted from the following block. <ul style="list-style-type: none"> A block in the G68 mode A block after G68 mode cancellation before execution of the X-/Z-axis positioning in the absolute data command 				
Action	<ol style="list-style-type: none"> Restart from a block not containing G06.2/G43.4 mode data. Modify the program so that the control of cross machining is canceled before another cross machining command is executed. Cancel the INTELLIGENT SAFETY SHIELD for automatic operation and execute EIA modal restarting search. Mount a milling tool and restart the program again or set 1 for bit 6 of parameter SU156. However, when 1 is set for bit 6 of parameter SU156, the return to the restarting position is not conducted. 				
957	MANUAL INTERRUPT NOT ALLOWED (WNo., NNo., BNo.)	E	I (L)	O	Red
Cause	Manual handle or MDI interruption from the block containing the G06.2 mode data has been attempted.				
Action	Perform manual interruptions only at blocks not containing G06.2 mode data.				
958	AUTO PECKING IMPOSSIBLE (WNo., NNo., BNo.)	E	I (L)	O	Red
Cause	<ol style="list-style-type: none"> The threshold value for load detection-based auto-pecking is not set to 0 or no such value is set. The parameter is not set appropriately. 				
Action	<ol style="list-style-type: none"> Set the appropriate threshold value either in the drill monitoring mode of the MACHINING-MONITORING display or on the TOOL DATA display. For parameter setting, contact Mazak Technical Center or Technology Center. 				
959	WORKPIECE COORDINATE ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The origin of the workpiece coordinate system does not lie on the axis of rotation of the table.				
Action	Review the settings on the WORK OFFSET display.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
960	SUPERPOSIT CTRL ILLEGAL COMMAND (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The program section under the mode of superposition control contains an incompatible G-code.				
Action	Review the program.				
961	ILLEGAL COMMAND 5X RADIUS COMP. (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> 1. A command was given which is incompatible with the current mode of tool radius compensation for 5-axis machining. 2. An arc having central angle of 180° or larger was specified in a circular interpolation command (G2/G3) in the tool radius compensation mode for 5-axis machining. 3. A rotary axis was specified in a circular interpolation command (G2/G3) in the tool radius compensation mode for 5-axis machining. 4. The 5-axis fairing function was specified in the tool radius compensation mode for 5-axis machining. 				
Action	<ol style="list-style-type: none"> 1. Cancel the tool radius compensation for 5-axis machining if the command in question (a G-code or T-code) must absolutely be given. 2. For the circular interpolation command (G2/G3) in the tool radius compensation mode for 5-axis machining, specify an arc with the central angle of 180° or larger by combining arcs which have the central angle smaller than 180°. 3. In the tool radius compensation mode for 5-axis machining, a rotary axis command cannot be specified in a circular interpolation command (G2/G3). 4. In the tool radius compensation mode for 5-axis machining, the 5-axis fairing function cannot be specified. 				
962	CANNOT USE 5X RADIUS COMP. (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	A command of tool radius compensation for five-axis machining has been designated under incompatible modal conditions.				
Action	Check the modal conditions for compatibility with the tool radius compensation for five-axis machining to cancel all the interlocking modes.				
963	TURRET MIRROR IMAGE N/A (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The optional mirror image function for the opposed turret is not provided.				
Action	Provide the NC unit with the mirror image option.				
	(, ,)				
Cause					
Action					
970	TOOL TIP CTRL PARAMETER ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The parameter settings on the composition of controlled axes for the following modes are not correct: tool tip point control, inclined-surface machining, tool radius compensation for five-axis machining, and workpiece placement error correction.				
Action	Correct the parameter settings concerned.				
971	CANNOT USE TOOL TIP PT CONTROL (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> 1. A command of tool tip point control is given under incompatible modal conditions. 2. A G43.5 command is given in spite of the selection of the workpiece coordinate system for programming. 3. Under selection of the G-code system for turning machines, a command of tool tip point control is given with the parameter setting "F161 bit 1 = 0 (Geometric offset by logically shifting the coordinate system)." 4. A command of circular interpolation is given under the mode of G43.5 or with the table coordinate system selected for programming. 				
Action	<ol style="list-style-type: none"> 1. Check the current modal conditions and cancel the unsuitable mode. 2. Use the other preparatory function G43.4, or select the table coordinate system for programming. 3. Set F161 bit 1 to "1" (Geometric offset by physically shifting the tool). 4. To use circular interpolation, give a G43.4 command, or select the workpiece coordinate system for programming. 				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
972	ILLEGAL CMD TOOL TIP PT CTRL (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	An unavailable command (of preparatory or tool function) is given in the mode of tool tip point control.				
Action	Cancel the mode of tool tip point control to give the required command with G- or T-code.				
973	ILLEGAL TOOL AXIS VECTOR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<p>An inappropriate command relevant to the vector of tool axis is given in the mode of tool tip point control.</p> <ol style="list-style-type: none"> In the mode of G43.4 with single-axis interpolation selected, the resulting tool path will not pass through the singular point although the sign of the angular position of the primary rotary axis is to be reversed. The command of the vector of tool axis is not correct (as it will reverse the direction of the tool). With single-axis interpolation selected, the command in a single block will cause a motion through more than 180° on the rotary axis of linear type. 				
Action	<ol style="list-style-type: none"> Modify the program for a tool path through the singular point, or select joint interpolation. Divide the command block concerned in order that the resulting rotation of the tool direction per block will not exceed 180°. Modify the program for a per-block motion angle of 180° or less on the rotary axis of linear type, or select joint interpolation. 				
974	TOOL TIP PT CTRL FORMAT ERROR (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	<ol style="list-style-type: none"> Arguments I, J, and K are specified in the mode of G43.4, or a command of rotary axis motion is given under G43.5. The code G49 is given in a block with other commands. A rotary axis is specified in the mode of circular interpolation. Under the modal condition of G43.4 or G43.5, an axis address other than those for the five controlled axes concerned (as specified in parameters K114, K115, K116, K121, and K125) was used in a motion command block. 				
Action	<ol style="list-style-type: none"> The attitude of the tool cannot be specified with I, J, and K in the G43.4 mode, nor can a rotary axis be explicitly specified in the G43.5 mode. Give the cancellation command G49 in a single-command block. A rotary axis cannot be specified in the mode of circular interpolation. Do not use an axis address other than those for the five controlled axes concerned (as specified in parameters K114, K115, K116, K121, and K125) in a motion command block under the modal condition of G43.4 or G43.5. 				
975	TOOL TIP PT CTRL N/A (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The optional function for tool tip point control is not provided.				
Action	The program with a command for tool tip point control can only be executed on a correspondingly executed machine.				
	(, ,)				
Cause					
Action					
979	MACRO USER ALARM (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = n (alarm message) in the user macroprogram was executed. n ≥ 21				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
980	MACRO USER ALARM 1 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 1 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
981	MACRO USER ALARM 2 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 2 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
982	MACRO USER ALARM 3 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 3 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
983	MACRO USER ALARM 4 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 4 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
984	MACRO USER ALARM 5 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 5 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
985	MACRO USER ALARM 6 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 6 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
986	MACRO USER ALARM 7 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 7 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
987	MACRO USER ALARM 8 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 8 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
988	MACRO USER ALARM 9 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 9 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
989	MACRO USER ALARM 10 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 10 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
990	MACRO MEASUREMENT ALARM 1 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	1. During execution of the MMS unit, the touch sensor has not come into contact with the workpiece (the skip signal has not turned on) when the maximum feed distance available at the skipping speed is exceeded. 2. #3000 = 11 (alarm message) in the user macroprogram was executed.				
Action	1. Check the machining program. 2. Refer to the relevant user macroprogram instruction manual to check the alarm.				
991	MACRO MEASUREMENT ALARM 2 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	1. During execution of the MMS unit, the touch sensor came into contact with the workpiece (the skip signal turned on) when another feeding than that at the skipping speed was taking place. 2. #3000 = 12 (alarm message) in the user macroprogram was executed.				
Action	1. Check the machining program. Also check the touch sensor for proper mounting on the spindle. 2. Refer to the relevant user macroprogram instruction manual to check the alarm.				
992	MACRO MEASUREMENT ALARM 3 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	1. Correct signals were not output because of trouble with the touch sensors, receivers or other such MMS unit components. 2. #3000 = 13 (alarm message) in the user macroprogram was executed. 3. An attempt has been made to perform MDI-MMS with the head not in its horizontal or vertical machining position.				
Action	1. Contact Mazak Technical Center or Technology Center. 2. Refer to the relevant user macroprogram instruction manual to check the alarm. 3. MDI-MMS is possible only with the head in its horizontal or vertical machining position.				
993	MACRO MEASUREMENT ALARM 4 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 14 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
994	MACRO MEASUREMENT ALARM 5 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 15 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
995	MACRO MEASUREMENT ALARM 6 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 16 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
996	MACRO MEASUREMENT ALARM 7 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 17 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
997	MACRO MEASUREMENT ALARM 8 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 18 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				

3

ALARM

No.	Message	Type of error	Stopped status	Clearing procedure	Display
998	MACRO MEASUREMENT ALARM 9 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 19 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				
999	MACRO MEASUREMENT ALARM 10 (, ,)	B	I (L)	O (S)	Red (Blue)
Cause	#3000 = 20 (alarm message) in the user macroprogram was executed.				
Action	Refer to the relevant user macroprogram instruction manual to check the alarm.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
1991	NOT POSSIBLE CROSS MACHINING (WNo., NNo., Cause)	B	I (L)	O (S)	Red (Blue)
Cause	<p>During the tool path check, the system in which the crossing command was set and/or the remote system for which the crossing command was to be executed has been set to the following status:</p> <ul style="list-style-type: none"> 1: The specified axis does not exist in the remote system. 2: The local system is specified as the system for which the crossing command was to be executed. 4: A crossing control command is set for the axis currently undergoing the crossing command. 16: The G110 command is set when the local system/remote system is in the modal environment that does not allow crossing control. 32: The crossing control command is set under the prohibited status of recalculation. 				
Action	Review and correct the program.				
1992	ILLEGAL AXES QTY. FOR CROSS (WNo., NNo., BNo.)	B	I (L)	O (S)	Red (Blue)
Cause	The number of axes in the remote system became zero during the tool path check when the crossing command was executed.				
Action	Review and correct the program.				
	(, ,)				
Cause					
Action					

3-2-11 No. 2000 - No. 2099 (3D setup error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display
2001	CAD FILE FORMAT ERROR (, ,)	A	L	S	Blue
<i>Cause</i>	Loading an incorrect CAD file has been attempted.				
<i>Action</i>	Load the correct CAD file.				
	(, ,)				
<i>Cause</i>					
<i>Action</i>					
2028	ILLEGAL WORKPIECE DIMENSION (, ,)	A	L	S	Blue
<i>Cause</i>	Incorrect dimensions have been entered in the material database or there are sections in which dimensions are missing.				
<i>Action</i>	Enter the correct dimensions.				
2029	ILLEGAL FIXTURE DIMENSION (, ,)	A	L	S	Blue
<i>Cause</i>	Incorrect dimensions have been entered in the fixture database or there are sections in which dimensions are missing.				
<i>Action</i>	Enter the correct dimensions.				
2030	CANNOT REGISTER (NUMBERS FULL) (, ,)	A	L	S	Blue
<i>Cause</i>	Despite 9999 sets of data already existing, registering further data has been attempted.				
<i>Action</i>	Up to 9999 sets of data can be registered. Delete unnecessary data to register desired one.				
2031	SELECT REGISTERED DATA (, ,)	A	L	S	Blue
<i>Cause</i>	Adding (copying) data has been attempted without selecting source data when editing the material database or the fixture database.				
<i>Action</i>	Select source data before adding (copying) data.				
2032	SELECT DATA TO CHANGE (, ,)	A	L	S	Blue
<i>Cause</i>	Changing data has been attempted without selecting source data when editing the material database or the fixture database.				
<i>Action</i>	Review the machining program.				
2033	SELECT DATA TO DELETE (, ,)	A	L	S	Blue
<i>Cause</i>	Deleting data has been attempted without selecting source data when editing the material database or the fixture database.				
<i>Action</i>	Select data before deleting the data.				
2034	CANNOT ADD WORKPCE MATERIAL (, ,)	A	L	S	Blue
<i>Cause</i>	Adding data to the material database has been attempted when an existing number was selected.				
<i>Action</i>	Specify a new number to add data.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
2035	CANNOT ADD FIXTURE (, ,)	A	L	S	Blue
Cause	Adding data to the fixture database has been attempted when an existing number was selected.				
Action	Specify a new number to add data.				
2036	SURFACE SELECTION ERROR (, ,)	A	L	S	Blue
Cause	Selecting the shape surface to be moved or disposed has been attempted during surface matching when a shape was not selected.				
Action	Select the shapes to be surface-matched.				
2037	DATA NOT REGISTERED (, ,)	A	L	S	Blue
Cause	Adding (copying) data or deleting data has been attempted when no data was present in the material database or the fixture database.				
Action	Add new data.				
2038	FILE WITH SAME NAME EXISTS (, ,)	A	L	S	Blue
Cause	The file name that has been entered during the name changing or copying process of model management is the same as the selected file name.				
Action	Enter a file name different from the selected file name.				
2039	FILE NAME TOO LONG (, ,)	A	L	S	Blue
Cause	Too long a file name has been entered during the model saving, renaming, or copying process of material model management or fixture model management.				
Action	Enter a file name in less than 256 characters.				
2040	WRITING ERROR (, ,)	A	L	S	Blue
Cause	The model saving process of material model management or fixture model management has failed.				
Action	Confirm the capacity of the HDD and save the model properly.				
	(, ,)				
Cause					
Action					
2049	NUMBER OF JAWS = 0 (, ,)	A	L	S	Blue
Cause	Displaying jaws has been attempted when the number of jaws was zero.				
Action	Specify one to nine jaws.				
2050	— (, ,)				
Cause	—				
Action	—				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
2051	MODEL SELECTION ERROR (, ,)	A	L	S	Blue
Cause	Parallel move surface matching, X-axis rotation surface matching, Y-axis rotation surface matching, or Z-axis rotation surface matching has been attempted when the shape to be moved or the surface of the shape to be moved was not selected.				
Action	Select the shape to be moved.				
2052	MODEL SELECTION ERROR (, ,)	A	L	S	Blue
Cause	Parallel move surface matching, X-axis rotation surface matching, Y-axis rotation surface matching, or Z-axis rotation surface matching has been attempted when the shape to be disposed or the surface of the shape to be disposed was not selected.				
Action	Select the shape to be disposed.				
2053	ILLEGAL SURFACE TYPE (, ,)	A	L	S	Blue
Cause	Parallel move surface matching, X-axis rotation surface matching, Y-axis rotation surface matching, or Z-axis rotation surface matching has been attempted when a surface other than a plane or cylindrical surface was selected.				
Action	Select a plane or a cylindrical surface.				
2054	ILLEGAL SURFACE ORIENTATION (, ,)	A	L	S	Blue
Cause	Parallel move surface matching has been attempted when surfaces that are not parallel were selected. X-axis rotation surface matching, Y-axis rotation surface matching, or Z-axis rotation surface matching has been attempted when a surface perpendicular to the rotational axis was selected.				
Action	Select surfaces whose relationship in position is correct.				
2055	NO PARALLEL PLANE (, ,)	A	L	S	Blue
Cause	The parallel surfaces to be moved in parallel for surface matching do not exist.				
Action	Parallel moving cannot be executed without parallel surfaces.				
2056	ILLEGAL FILE NAME (, ,)	A	L	S	Blue
Cause	Illegal characters or symbols are included in the file name.				
Action	Enter the characters and symbols that can be used in the file name.				
2057	NUMBER ALREADY REGISTERED (, ,)	A	L	S	Blue
Cause	A registered number has been designated during database registration.				
Action	Designate a new number.				
	(, ,)				
Cause					
Action					
2061	WORK MATL. LARGER THAN BARRIER (, ,)	A	L	S	Blue
Cause	The specified parametric model or CAD model is outside the workpiece information of the common unit.				
Action	Dispose the parametric model or the CAD model so as to stay internally to the workpiece information. Even if the model is outside the information, the specification itself of that model is possible.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
2062	Fixture larger than barrier (, ,)	A	L	S	Blue
Cause	The specified CAD model is outside the jaw shape based on the setup information.				
Action	Dispose the CAD model so as to stay internally to the setup information. Even if the model is outside the information, the specification itself of that model is possible.				
2063	Cannot move model (, ,)	A	L	S	Blue
Cause	It is not possible to manually dispose a material model that has been created from the workpiece information, or a fixture model that has been created from setup information.				
Action	Modify the Z-offset or WPC-Z to move the material model. Modify the settings of BA19 and BA20 to move the fixture model.				
2064	— (, ,)				
Cause	—				
Action	—				
2065	NO 2ND SPINDLE (, ,)	A	L	S	Blue
Cause	The menu function for the secondary spindle has been selected under machine specifications not having the secondary spindle.				
Action	The menu function for the secondary spindle can be used only under the machine specifications not having the secondary spindle.				
	(, ,)				
Cause					
Action					

3-2-12 No. 2100 - No. 2199 (Interference error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display
2100	MACN INTRF (WNo., UNo., SNo.)	A	L	S	Blue
Cause	Execution of the machining program results in the interference occurring between <Interfering section 1> and <Interfering section 2>.				
Action	Review and correct the program.				
2101	NEAR MISS (WNo., UNo., SNo.)	A	L	S	Blue
Cause	Execution of the machining program results in <Interfering section 1> and <Interfering section 2> being present within the specified interference distance.				
Action	Review and correct the program.				
2102	WORK INTRF (WNo., UNo., SNo.)	A	L	S	Blue
Cause	Execution of the machining program results in the interference occurring between <Interfering section 1> and <Interfering section 2>.				
Action	Review and correct the program.				
	(, ,)				
Cause					
Action					
2110	ILLEGAL FORMAT (, ,)	A	L	S	Blue
Cause	The entered data is incorrect.				
Action	Review the range of the data to be entered.				
2111	NO SOLID CHECK IN AUTO OPER (, ,)	A	L	S	Blue
Cause	Execution of Virtual Machining has been attempted during automatic operation of the NEXUS machine.				
Action	Terminate automatic operation before executing Virtual Machining.				
2112	Z-OFFSET NOT FOUND (WNo., , ,)	B	L	S	Blue
Cause	A Z-offset is not set in the program of Z-offset scheme.				
Action	Set the Z-offset.				
2113	JAW NO NOT FOUND (WNo., , ,)	B	L	S	Blue
Cause	Jaw No. is not set.				
Action	Set Jaw No.				
	(, ,)				
Cause					
Action					

No.	Message	Type of error	Stopped status	Clearing procedure	Display
2120	RESTART PT SEARCH INTERRUPTED (, ,)	A	L	S	Blue
Cause	The restart position search function has been aborted.				
Action					
2121	RESTART POINT NOT FOUND (, ,)	A	L	S	Blue
Cause	The restarting point is not found.				
Action	Review the restrictions on the restart or review the machining program itself.				
	(, ,)				
Cause					
Action					
2130	CANNOT CHECK WORKPIECE MODEL (, ,)	A	L	S	Blue
Cause	1. Lack of available memory space 2. Arithmetical incapability 3. Entered data imperfections 4. A program containing a chain from a MULTIPLEX program to a non-MULTIPLEX program has been checked in the solid mode. (Only for MULTIPLEX series)				
Action	1. Divide the program. 2. Confirm parameters. 3. Confirm tool data. 4. Review the program.				
	(, ,)				
Cause					
Action					
2140	CANNOT DISPLAY MACHINE MODEL (, ,)	A	L	S	Blue
Cause	1. Lack of available memory space 2. Arithmetical incapability 3. The machine model file is missing.				
Action	1. Divide the program. 2. Confirm parameters. 3. Contact Mazak Technical Center or Technology Center.				
	(, ,)				
Cause					
Action					
2150	CANNOT DISPLAY TOOL MODEL (, ,)	A	L	S	Blue
Cause	The shape cannot be created since the entered tool data is insufficient or incorrect.				
Action	Review and correct the tool data.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
2151	CANNOT DISPLAY TAILSTOCK MODEL (, ,)	A	L	S	Blue
Cause	The shape cannot be created since the entered machine parameters on the tailstock shape are incorrect.				
Action	Modify the data settings of machine parameters BA12 to BA14 .				
	(, ,)				
Cause					
Action					
2190	NO MACHINE MODEL (, ,)	B	L	N	Blue
Cause	A machine model is not registered.				
Action	Register a machine model and then turn power back on.				
2191	CANNOT SELECT CHECK UNITS (, ,)	B	L	Q	Blue
Cause	In spite of the setting for a material piece to be mounted on the machine table, it was attempted to selectively turn off the function of checking for interference with respect to the material piece.				
Action	Clear the setup setting for the material piece in question beforehand.				
2192	CHECK UNITS SET ERROR (, ,)	B	L	Q	Blue
Cause	A machining program is searched for, or selected, for which a material piece is arranged in a position where the function of checking for interference is currently specified as being cancelled for the material piece.				
Action	Turn on the function of interference check for the material piece in the particular mounting position.				
	(, ,)				
Cause					
Action					

3-2-13 No. 2200 - No. 2299 (CAM data output error)

No.	Message	Type of error	Stopped status	Clearing procedure	Display
2201	READ-ONLY MEDIA (, ,)	G	L	S	Blue
Cause	The external memory is write-protected.				
Action	Remove the write protection from it.				
2202	NETWORK LOGON FAILED (, ,)	A	L	S	Blue
Cause	A user failed to log on to another PC when storing data on it.				
Action	Check if the user in question is allowed to log on to the PC specified as the destination device.				
2203	NETWORK PATH NOT FOUND (, ,)	A	L	S	Blue
Cause	The folder specified as the destination folder is not found on the network.				
Action	Check again if the destination folder on the network is correctly specified.				
2204	HARDWARE NOT INSTALLED (, ,)	A	L	S	Blue
Cause	The memory device is not inserted correctly.				
Action	Insert the device correctly and make a retry.				
2205	CANNOT ACCESS FILE (, ,)	A	L	S	Blue
Cause	The user does not have the authority to access the PC specified.				
Action	Check if the user in question has the authority to access the PC specified.				
2206	FILE CONFIGURATION ERROR (, ,)	E	L	S	Blue
Cause	The file is not found or it is corrupted.				
Action	Contact Mazak Technical Center or Technology Center.				
2207	CANNOT ACCESS CNC (, ,)	E	L	S	Blue
Cause	A communication error occurred between the NC and the PC.				
Action	Check if NC displays can be smoothly switched from one to another. If this alarm is displayed again after the NC has been turned off and back on, check what operation was performed when the error occurred and also the value displayed in (, ,), and then contact Mazak Technical Center or Technology Center.				
2208	MACHINE DATA TRANSFER ERROR (, ,)	E	L	S	Blue
Cause	An error occurred while machine data was being transferred.				
Action	Make a retry. If this alarm is displayed again after a retry, check what operation was carried out when the error was occurred and also the value displayed in (, ,), and then contact Mazak Technical Center or Technology Center.				

No.	Message	Type of error	Stopped status	Clearing procedure	Display
2209	CANNOT SAVE MACHINE DATA (Note, ,)	G	L	S	Blue
Cause	Note : Sub error codes related to the storage of machine data 300 to 399: Error in storing data on the destination device caused by lack of storage space 400 to 499: Error in extracting or compressing machine data				
Action	For sub error codes of 300's: Check if there is enough free space in the memory device, and if the device is correctly recognized. For sub error codes of 400's: Check if there is enough free space on the hard disk and make a retry if there is no problem. If this alarm is displayed again after a retry, check what operation was carried out when the error was occurred and also the value displayed in (, ,), and then contact Mazak Technical Center or Technology Center.				
2210	INVALID DRIVE SPECIFIED (, ,)	E	L	S	Blue
Cause	An invalid drive is specified.				
Action	Specify a drive available.				
2211	FILE SAVINGFAILURE (, ,)	G	L	S	Blue
Cause	A user failed to write data on the destination device specified.				
Action	Check if you have the authority to write data to the destination. If the authority is present, check the presence or absence of invalid folders or files, the free space left in the destination device, and if the device is correctly recognized.				
2212	FILE NOT FOUND (, ,)	B	L	S	Blue
Cause	Some are missing from the files to be saved as CAM machine data.				
Action	The data you want to copy may be corrupted. If this alarm is displayed again after a retry, check what operation was carried out when the error was occurred and also the value displayed in (, ,), and then contact Mazak Technical Center or Technology Center.				
	(, ,)				
Cause					
Action					

4 M-CODE LIST

M-code	Function
M00	Programmed stop Stops reading and execution of a program; the P. STOP in the CNC Signal window goes on. At this time, the spindle will also stop rotating.
M01	Optional stop Executes the same function as the programmed stop function (M00) provided that the [OPTIONAL STOP] menu function is selected. The M01 command is ignored if the [OPTIONAL STOP] menu function is not selected.
M02	End of program Stops the spindle, coolant, air blast and oil mist. The NC functions are all reset and the machine stops. Except in tape operation mode, a function similar to that of M30 is performed.
M03	Spindle normal rotation Starts the spindle rotation in the normal direction.
M04	Spindle reverse rotation Starts the spindle rotation in the reverse direction.
M05	Spindle stop Stops spindle rotation started by M03 or M04.
M06	Tool change Mounts the specified tool in the spindle and returns the tool of the spindle to the magazine. If the next tool command is specified, the next tool is loaded from the magazine to the ATC arm. Sample program: 1. M06 <u>T**</u> <u>TΔΔ</u> ↑ ↑ Specified tool Next tool 2. T**; M06; TΔΔ M06; Machining program 3. M06 <u>T**</u> ↑ Valid/invalid is selected by parameter setting. (Standard: Invalid)
M07	Mist coolant ON (Optional) Starts the oil-mist coolant system.
M08	Flood coolant ON Starts the flood coolant system to discharge coolant from the nozzles.
M09	All coolant OFF Turns off all coolant systems and air blast system. It is not allowed to specify a coolant supply M-code preceding M09 in the same block.
M10	Tool clamp Clamps a tool.
M11	Tool unclamp Unclamps a tool. For this operation, the spindle must be stopped in the oriented position.
M17	Tool unload Moves the ATC shifter to the magazine. For this operation, the magazine must be indexed correctly.
M18	Tool load Moves the ATC shifter to the spindle. For this operation, the magazine must be indexed correctly.
M19	Spindle orientation Stops the spindle in the oriented stop position (tool change angular position).

M-code	Function
M23	Error detect ON Selects the exact stop mode. For details of the exact stop mode, refer to the Operating Manual for the NC unit.
M24	Error detect OFF Cancels the M23 mode.
M30	Reset and rewind Stops the machine similarly to M02, resets the function of the NC unit, and returns control to the beginning of the program.
M33	Tool length measuring device extend (Optional) Sets the tool length measuring device to the measurement position for tool breakage detection or tool length measurement.
M34	Tool length measuring device retract (Optional) Returns the tool length measuring device to the standby position for tool breakage detection or tool length measurement cycle.
M35	Tool breakage detection (Valid only for MAZATROL programs) (option) Calls the tool breakage detection cycle. In the first ATC cycle after the execution of the M35 command, the selected tool is checked for breakage. For details of this function, refer to the Operating Manual of the NC unit.
M42	Index table reverse rotation Rotates the index table in reverse. Use this M-code with an argument B for indexing the table by a reverse turn.
M46	4th axis table unclamp (Optional) Keeps the 4th-axis table unclamped. Use this M-code for contouring, for example, by continuous cutting feed on the 4th axis.
M47	4th axis table clamp (Optional) Clamps the 4th-axis table.
M48	M49 cancel Cancels the override cancel function which is specified by M49. In the M48 mode, the override function is valid.
M49	Override cancel Cancels the override function (for feed rate and spindle speed). Therefore, the machine operates with programmed data. Note : When power is switched on or after emergency stop, the NC is (re)set in the M48 mode.
M50	Air blast ON (Optional) Starts the air blast system. To stop the air blast system, execute M09.
M51	Through-spindle coolant ON (Optional) Discharges coolant through the through-hole in the tool if oil-hole attachment is used. To stop the air blast system, execute M09.
M52	Tap coolant ON (Optional) Discharges tap coolant if the tap coolant unit is installed. To stop the tap coolant, execute M09.
M58	Tool life check Brings the machine to a single-block stop if an alarm is displayed to indicate that the life of the tool mounted in the spindle has expired. In this case, the operation of the machine can be continued by pressing the cycle start button after pressing the clear key  .
M68	Pallet clamp Clamps a pallet.
M69	Pallet unclamp Unclamps a pallet.
M71	Pallet No. 1 selection Selects pallet No. 1 for pallet change operation.

M-code	Function
M72	Pallet No. 2 selection Selects pallet No. 2 for pallet change operation.
M73	Pallet No. 3 selection (Optional) Selects pallet No. 3 for pallet change operation (for the machine with 6PC).
M74	Pallet No. 4 selection (Optional) Selects pallet No. 4 for pallet change operation (for the machine with 6PC).
M75	Pallet No. 5 selection (Optional) Selects pallet No. 5 for pallet change operation (for the machine with 6PC).
M76	Pallet No. 6 selection (Optional) Selects pallet No. 6 for pallet change operation (for the machine with 6PC).
M90	Mirror image cancel Cancels the mirror image function called by M91, M92, or M93.
M91	X-axis mirror image ON Makes mirror image for the X-axis valid. When this function is active, axis motion direction is reversed for the X-axis.
M92	Y-axis mirror image ON Makes mirror image for the Y-axis valid. When this function is active, axis motion direction is reversed for the Y-axis.
M93	B-axis (4th) mirror image ON (Optional) Makes mirror image for the 4th axis valid. Reverses the sign (plus, minus) for the rotation amount of the table (4th axis).
M98	Subprogram call Calls a subprogram. For details of this function, refer to the Operating Manual for the NC.
M99	Return to main program Indicates the end of a subprogram. For details of this function, refer to the Operating Manual for the NC.
M119	Multi-point orientation (optional) Orients the spindle at the desired angular position when specified with an S code in the same block. It is specified in the format of "M119 S***". Note : "S***" specifies the angular position of the spindle in the range from 0 to 359 in 1° units.
M120	Automatic power shut-off Shuts off the main power to the machine if the AUTO POWER OFF switch is set to the "ON" position.
M122	Gap eliminator ON (Optional) Selects the function to increase the rate of feed during air cutting by multiplying the programmed rate by the parameter-set value to reduce the air cut time.
M123	Gap eliminator OFF (Optional) When the power is switched on, the gap eliminator function is OFF. This M-code is used to cancel the gap eliminator function selected by M122.
M130	NIAGARA coolant ON (Optional) Starts the NIAGARA coolant system if equipped.
M132	Through-spindle air ON (Optional) Discharges through-spindle air.
M134	IN station pallet ID read (Optional) Reads the pallet data stored in the data carrier on the pallet placed at the IN station.
M137	IN station pallet ID write (Optional) Writes the pallet data to the data carrier on the pallet placed at the IN station.
M138	OUT station pallet all ID data write (Optional) Writes all the pallet data to the data carrier on the pallet placed at the OUT station.
M139	Heavy tool mode cancel Cancels the heavy tool mode.
M140	Heavy tool mode designation Designates the heavy tool mode.

M-code	Function
M149	Magazine indexing Indexes the magazine for the specified tool. Give an M149 T $\Delta\Delta$ command to select the tool No. $\Delta\Delta$.
M155	Thermal displacement compensation cancel Cancels the function of thermal displacement compensation. The M155 mode is cancelled by resetting, M06 (tool change), or switching-off.
M160	M161, M162 cancel Sets the spindle speed arrived completion level at 85% of the specified spindle speed.
M161	Spindle speed arrived setting at 70% Sets the spindle speed arrived completion level at 70% of the specified spindle speed.
M162	Spindle speed arrived setting at 50% Sets the spindle speed arrived completion level at 50% of the specified spindle speed.
M169	M170 cancel Makes the AFC function valid. (AFC is usually valid.)
M170	AFC cancel Cancels the AFC function.
M173	Dynamic offsetting ON (Optional) Selects the dynamic offsetting function (for the NC rotary table).
M174	M173 cancel Cancels the dynamic offsetting function.
M178	Zero point return for the index table Returns the index table to the 0° position.
M179	Zero point return for the magazine Returns the magazine to the indexing reference position (for pocket No. 1).
M180	M181 cancel Cancels the function called out by M181.
M181	Next tool indexing invalid Neglects the next tool command in order to prevent an evil effect of the vibration associated with magazine or ATC shifter operation on the surface finish.
M195	Tool breakage detection ON (option) Executes a tool breakage detection cycle. This command is automatically specified right before the ATC cycle when M35 is entered in a MAZATROL program. Usually, this command is not used by the customer.
M196 M197	Fully-automatic tool length measurement mode ON (Optional) Fully-automatic tool length measurement mode OFF (Optional) After the execution of M196, automatic tool length measurement cycle (mounting the specified tool to the spindle, measuring the tool's length, and registering the measurement result) is carried out in response to each T-code until the M196 mode is canceled by M197 or resetting. In this fully-automatic mode, it is not possible to measure the length of tools such as a face milling cutter which do not have their nose on the center line. These M-codes are automatically outputted when "automatic tool measurement [T MSR AUTO]" is selected in the MDI mode. M197 includes the retraction of the measuring device.
M198	Semi-automatic tool length measurement (Optional) In this mode, the length of the tool presently mounted in the spindle is measured. Since the measurement cycle starts from the present position, this cycle can be used to measure the length of tools which do not have their nose on the center line. This M-code is automatically outputted when "semi-automatic tool measurement [T MSR SEMI AUTO]" is selected in the MDI mode.
M199	Fully-automatic tool length measurement (Optional) In this mode, the length of the tool presently mounted in the spindle is measured.
M380	NIAGARA coolant on the spindle side ON Starts NIAGARA coolant feed on the spindle side.
M381	NIAGARA coolant on the pallet changer side ON Starts NIAGARA coolant feed on the pallet changer side.

Note 1: Simultaneous processing impossible M-codes

Up to 4 M-codes can be entered in one block. However, if M-codes which cannot be processed simultaneously are entered in the same block, the following message is displayed and operation is suspended.

227 SIMULTANEOUS M CODE OPERATION

This alarm occurs in the following cases:

- Any combination of M03, M04, M05 and M19
 - M23 and M24 (opposing operation)
 - M33 and M34 (opposing operation)
 - M48 and M49 (opposing operation)
 - M15 and M33 (opposing operation)
 - M06 and M149 (opposing operation)
 - Any combination of M71 to M80 (for pallet changer specification)
 - M195 to M198, specified with other M-code

Note 2: When entering more than one M-code in the same block, there is a restriction on entry order when entering M-codes related with coolant and air blast system (M09, M07, M08, M50, M51, M52, M130, M131, M132, etc.).

An M-code starting the coolant system or air blast system must not be entered before M09.

To enter M08 before M09, for example, enter M08 in a block preceding the M09 block.

M08, M09, M51, → M08
M09, M51,



Copyright(c) 2013 YAMAZAKI MAZAK CORPORATION. All rights reserved.
Serial No. 275288